

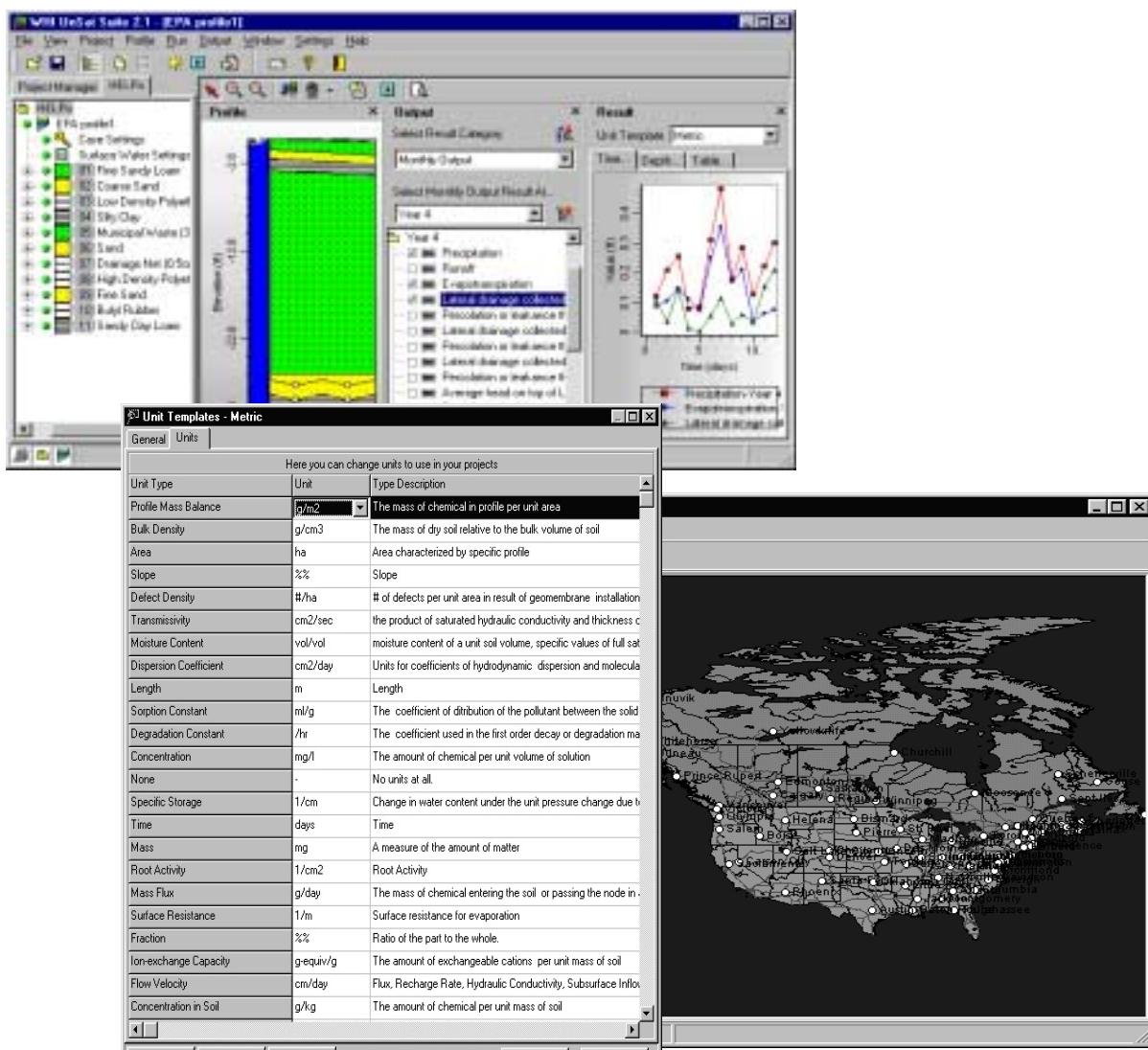
User's Manual

for

WHI UnSat Suite

(includes Visual HELP)

The intuitive unsaturated zone analysis package



Developed by: Waterloo Hydrogeologic, Inc.

License Agreement

Waterloo Hydrogeologic Inc. retains the ownership of this copy of the software. This copy is licensed to you for use under the following conditions:

I. Copyright Notice

This software is protected by both Canadian copyright law and international treaty provisions. Therefore, you must treat this software JUST LIKE A BOOK, with the following single exception. Waterloo Hydrogeologic Inc. authorizes you to make archive copies of the software for the sole purpose of backing-up our software and protecting your investment from loss.

By saying "JUST LIKE A BOOK", Waterloo Hydrogeologic Inc. means, for example, that this software may be used by any number of people within the organization that purchased the software, and it may be freely moved from one computer location within the licensed organization to another, so long as there is NO POSSIBILITY of it being used at one location while it is being used at another. Just like a book can't be read by two different people in two different places at the same time.

Specifically, you may not distribute, rent, sub-license, or lease the software or documentation; alter, modify, or adapt the software or documentation, including, but not limited to, translating, decompiling, disassembling, or creating derivative works without the prior written consent of Waterloo Hydrogeologic Inc. The provided software and documentation contain trade secrets and it is agreed by the licensee that these trade secrets will not be disclosed to non-licensed persons without written consent of Waterloo Hydrogeologic Inc.

II. Warranty

Waterloo Hydrogeologic Inc. warrants that, under normal use, the material of the magnetic diskettes and the documentation will be free of defects in materials and workmanship for a period of 30 days from the date of purchase. In the event of notification of defects in material or workmanship, Waterloo Hydrogeologic Inc. will replace the defective diskettes or documentation.

The remedy for breach of this warranty shall be limited to replacement and shall not encompass any other damages, including but not limited to loss of profit, and special, incidental, consequential, or other similar claims.

III. Disclaimer

Except as specifically provided above, neither the developer(s) of this software nor any person or organization acting on behalf of him (them) makes any warranty, express or implied, with respect to this software. In no event will Waterloo Hydrogeologic Inc. assume any liabilities with respect to the use, or misuse, of this software, or the interpretation, or misinterpretation, of any results obtained from this software, or for direct, indirect, special, incidental, or consequential damages resulting from the use of this software.

Specifically, Waterloo Hydrogeologic Inc. is not responsible for any costs including, but not limited to, those incurred as a result of lost profits or revenue, loss of use of the computer program, loss of data, the costs of recovering such programs or data, the cost of any substitute program, claims by third parties, or for other similar costs. In no case shall Waterloo Hydrogeologic Inc.'s liability exceed the amount of the license fee.

IV. Infringement Protection

Waterloo Hydrogeologic Inc. is the sole owner of this software. Waterloo Hydrogeologic Inc. warrants that neither the software and documentation nor any component, including elements provided by others and incorporated into the software and documentation, infringes upon or violates any patent, trademark, copyright, trade secret, or other proprietary right.

Royalties or other charges for any patent, trademark, copyright, trade secret or other proprietary information to be used in the software and documentation shall be considered as included in the contract price.

V. Governing Law

This license agreement shall be construed, interpreted, and governed by the laws of the Province of Ontario, Canada, and the United States. Any terms or conditions of this agreement found to be unenforceable, illegal, or contrary to public policy in any jurisdiction will be deleted, but will not affect the remaining terms and conditions of the agreement.

VI. Entire Agreement

This agreement constitutes the entire agreement between you and Waterloo Hydrogeologic Inc.

Table of Contents

UnSat Suite Interface.....	1
Introduction	3
WHI UnSat Suite Models	3
History	4
New Features of WHI UnSat Suite 2.2.....	4
The WHI UnSat Suite Concept.....	5
How to Contact WHI	6
Software Maintenance and Technical Support.....	7
Other Products by WHI	7
Waterloo Hydrogeologic Inc. Training and Consulting.....	8
Acknowledgments	8
1. Installation and Start-Up	11
System Requirements	11
Installing WHI UnSat Suite 2.2 from the CD-Rom	11
Starting WHI UnSat Suite.....	12
2. General Features	13
Terms and Notation	13
Getting Around the Interface	13
Main Menu	14
Operational Icons	15
Profile Icons	16
Project Tree View.....	17
Profile View	17
Output View.....	17
Results View	17
Starting a New Project	17
General Project Information.....	18
Selecting a Problem and Model	18
Location	19
Units	23
Authors and Clients	26
Loading an Existing Project	28
Profiles in UnSat Suite	28
Creating a New Profile Using a Profile Template	30
Creating a New Profile	30
Creating a New Profile Template	32
Starting the Simulation	32
Using the Material Designer.....	32
Adding new contaminant to the database.....	35

Modifying and Deleting Materials	35
3. Common Tools	37
Working in the Project Tree View	37
Elements of the Profile Structure	37
Indicators of the Object's Status.....	39
Accessing and Editing Objects of Profile Structure.....	39
Working in the Profile View.....	44
Merging Layers.....	45
Restoring a Layer	47
Splitting a Layer	47
Resizing the Layer	49
Working with Multiple Profiles	49
Running the Model	50
Viewing the Output.....	50
Viewing the Output Graphs.....	51
Viewing Tables	55
Editing Graphs.....	56
Printing an Individual Graph.....	58
Preparing a Report.....	59
4. Working with Project Sets	61
Definition of the Project Set	61
Using the Project Set Manager	61
Deleting Alias.....	63
Archiving and Copying Project Sets.....	63
Returning to the Default Project Set.....	64
Deleting Projects from the Default Project Set.....	64
Opening an Archive Project Set	65
Opening a Local Project Set.....	65
Working over a Local Area Network	65
Copying a Project From One Project Set to Another	67
Repairing the Project Set.....	68
The HELP Model	69
Introduction.....	71
History.....	71
New Features of Visual HELP 2.1	72
New Features of Visual HELP 2.2	72
Importing a Visual Help 1.1 model into Visual Help 2.2.....	72
5. Designing the Landfill Profile.....	75
Profiles in Visual HELP	75
Subprofiles	77

Layering Rules	78
Profile Properties	78
Case Settings	79
Editing Case Settings	80
Runoff Method	80
Initial Moisture Settings	81
Editing the Surface Water Settings	82
Editing Landfill Layers and Modifying the Profile	82
Layer Properties	83
Editing Layer Properties	85
Resizing Layers	87
Inserting Layers	89
Deleting Layers	91
Restoring Layers	93
Splitting Layers	94
Layer Groups	96
6. Generating Weather Data	99
Introduction	99
World Weather Generator Database	100
Starting the Weather Generator	101
Getting Around	102
Precipitation and Temperature	111
Evapotranspiration	112
Generating Weather Data	114
Viewing Generated Weather Data	115
Editing Generated Data	118
Viewing the Weather Generator Database	120
Adding a Record to the Weather Generator Database	121
Editing the Weather Generator Database	122
Importing Weather Data in Canadian Climate Centre Format	123
Importing Weather Data in NOAA format	124
Customizing Weather Data	127
7. Running the Model, Viewing Output, and Reporting	133
Setting the Simulation Time with the Weather Generator	133
Running the Visual HELP Model	134
Interpreting the Output and Preparing a Report	134
Original DOS HELP Output	134
Viewing the Output Graphs	135
Viewing HELP Tables	139
Explanation of Variables	141
Creating a Report	142
The PESTAN Model	145

Introduction	147
8. Input Specification.....	149
The Case Settings	149
Editing Sorption Constant.....	150
Time Dependent Groups	150
Waste Application Schedule.....	151
Observation Times	152
Specifying the Pesticides.....	153
Substituting the Pesticide	153
Editing Pesticide Properties	154
Modifying the Profile	155
Profile Properties	155
Resizing the Layer	155
Substituting the Layer.....	156
Editing Soil Properties	157
Observation Point	159
Adding Observation Points.....	159
Observation Point Properties	160
Deleting an Observation Point	161
Restoring an Observation Point	161
9. Viewing Output and Reporting	163
Original DOS PESTAN Output	163
Viewing the Output Graphs.....	163
Specified Time and Depth	164
Observation Point.....	167
Balance	167
Viewing Tables	168
Creating a Report	169
The VS2DT Model	171
10. Introduction	173
11. Case Specifications.....	175
Case Settings	175
Transport Simulation	176
Soil Hydraulic Function	178
Initial Conditions	179
Maximum Simulation Time.....	180
Evapotranspiration	180
Solver Settings.....	181

Editing the Solver Settings	181
Evapotranspiration Parameters	183
Editing the Evapotranspiration Parameters	184
Boundary Conditions	186
Boundary Rules	188
Flow Boundaries	188
Transport Boundaries	190
Stress Period Defaults	192
Specifying the Stress Period Defaults	192
Setting Output Times	193
Observation Times	193
The Finite Difference Grid	195
Customizing the Finite Difference Grid	196
Observation Points	197
Adding Observation Points	197
Observation Point Properties	199
Deleting an Observation Point	199
Restoring an Observation Point	200
Setting the Initial Conditions	200
Selecting the Type of Initial Condition	200
Editing the Initial Conditions for a Layer	201
12. Modifying the Profile	203
Profile Properties	203
Changing the Soil Profile Layer Structure	204
Merging Layers and Erasing Layer Boundaries	205
Restoring a Layer	206
Splitting a Layer	207
Resizing a Layer	208
Changing Properties of a Layer	209
Substituting Material in a Layer	210
Editing Soil Hydrologic Properties	210
Editing the Soil Parameters	210
Permanent Soil Hydrologic Parameters	211
Dependent Soil Parameters	212
Transport Parameters	212
Editing Transport Parameters	213
Permanent Soil Hydrologic Parameters	215
Dependent Transport Parameters	215
13. Running the Model, Viewing Output, and Reporting	217
Running the VS2DT Model	217
Viewing the Original VS2DT Input and Output Files	217
Viewing the Output Graphs	218
Specified Time and Depth	219
Balance, Accumulated Balance and Rate	222

Viewing Tables	223
Creating a Report	224
Editing Model Stress Periods	225
The VLEACH Model.....	227
14. Introduction	229
15. Input Specification.....	231
Specifying the Case Settings.....	231
Specifying the Contaminant	232
Setting Initial Conditions	232
Substituting the Contaminant.....	234
Editing Contaminant Properties	235
Modifying the Profile	235
Profile Properties	235
Resizing the Layer	237
Substituting the Layer.....	237
Editing Soil Properties	238
16. Viewing Output and Reporting	241
Original DOS VLEACH Output.....	241
Viewing the Output Graphs.....	242
Specified Time and Depth	242
Balance	245
Viewing Tables	246
Preparing a Report.....	248
The SESOIL Model	251
17. Introduction	253
18. Input Specification.....	255
Profiles in SESOIL	255
Editing General Profile Properties	256
Specifying the Case Settings.....	256
Specifying Climate	258
Inputting climate data manually	260
Input climate data from the Weather Generator database	262
Input weather data synthetically generated in SESOIL	262
Import synthetically generated weather data and evapotranspiration from the HELP model	263

Specifying Soil Erosion and Contaminant Washload	265
Specifying Washload Settings	265
Specifying Washload Schedule	266
Specifying the Contaminant	267
Defining the Contaminant	267
Editing Chemical Properties	268
Setting Initial Conditions	270
Contaminant Application Schedule	271
Modifying the Profile	273
Profile Properties	273
Setting the Profile Material	274
Setting Layer Structure of the Profile	276
Setting Groundwater Parameters	277
19. Running the Model, Viewing Output and Reporting	279
Running the SESOIL Model	279
Viewing Original DOS SESOIL Output	279
Viewing the Output Graphs	281
Specified Depth (Annual Summary)	282
Viewing Specified Depth (Monthly Output)	289
Specified Time (Concentration)	291
Preparing a Report	294
Export and Internal Transfer of Simulation Results	297
20. Export , Internal Transfer, and Import of Simulation Results	299
Internal Data Transfer between WHI UnSat Suite Models	299
Export from WHI UnSat Suite	309
Export of Visual HELP data to Visual MODFLOW	309
Export of SESOIL data to Visual MODFLOW	315
Export of VS2DT data to Visual MODFLOW	327
Export of VS2DT data to Visual MODFLOW for multiple species	329

Part 1:

UnSat Suite Interface

Introduction

The WHI UnSat Suite combines models HELP, PESTAN, SESOIL, VLEACH and VS2DT in powerful new graphical environment specifically designed for simulating one-dimensional groundwater flow and contaminant transport through the unsaturated zone. All five of these popular models are seamlessly integrated with the WHI UnSat Suite and each one is compiled and optimized to run as a 32-Bit, native Windows application.

WHI UnSat Suite Models

HELP is a versatile U.S. EPA model for predicting landfill hydrologic processes and testing the effectiveness of landfill designs, enabling the prediction of landfill design feasibility. HELP has become a requirement for obtaining landfill operation permits in the U.S.A. HELP is also effective in estimating of groundwater recharge rates.

PESTAN. The PESTAN (PESTicide ANalytical) model uses an analytical solution to predict the transport of organic solutes through the unsaturated zone to the groundwater table. The model is commonly used for initial screening assessments to evaluate the potential for groundwater contamination by pesticides used in agricultural applications. However, it is also quite useful for determining potential groundwater impacts from any organic solutes migrating through the unsaturated zone.

SESOIL is a popular US EPA model which is capable to simultaneously model water transport, sediment transport and pollutant fate. SESOIL is widely used by consultants and state regulatory agencies as a screening tool to assess contaminant fate and transport for regulatory requirements. The ability of the model to account for contaminant washload, volatilization and air diffusion of the volatile organic contaminants, as well as for sorption, volatilization, degradation, cation exchange, hydrolysis and metal complexation makes it a unique tool which only can be applied in many practical cases.

VS2DT is a finite difference numerical model for simulating steady-state or transient, Variably-Saturated 2-D groundwater flow and solute Transport. Typical applications of the VS2DT model include determining the fate of agricultural chemicals, landfill leachate, UST leaks, and accidental chemical spills as they migrate through the unsaturated zone towards the water table.

VLEACH is a one-dimensional finite difference Vadose zone LEACHing model for predicting the vertical mobilization and migration of organic contaminants in the vadose zone. This model is commonly used to evaluate groundwater impacts due to vertical migration of organic

contaminants through the unsaturated zone, and to predict subsurface volatilization of VOCs.

History

WHI has developed a Windows interface for the version 3.08 of the HELP model which was released in May 1998 under the name Visual HELP version 1.101. This software was presented at the trade show at SWANA's WASTECON 1998 and ISWA World Congress 1998 in Charlotte, North Carolina.

WHI UnSat Suite was released in May 1999. The product combined models HELP, PESTAN, VLEACH and VS2DT in a powerful new graphical environment specifically designed for simulating one-dimensional groundwater flow and contaminant transport through the unsaturated zone. The ideas and approaches tested in Visual HELP version 1.101 have been advanced. The product has been sold to our clients in three versions:

Visual HELP 2.1 which included the HELP model,

WHI UnSat Suite 2.1 which included PESTAN, VLEACH and VS2DT, and

WHI UnSat Suite Plus 2.1 which included HELP, PESTAN, VLEACH and VS2DT.

Currently, more than 400 copies of this product are being used by consulting companies, government regulator bodies and Universities in U.S.A., Canada, U.K., Germany, Australia, Sweden, Mexico, France, Slovenia, Slovakia and Hungary.

New Features of WHI UnSat Suite 2.2

The SESOIL Model has been added

SESOIL, a popular US EPA model which is capable to simultaneously model water transport, pollutant washload, sediment transport and pollutant fate has been added to the modelling suite.

The Pollutant Parameter Database has been expanded

The database of pollutant parameters for PESTAN model has been expanded. The new pollutant database for SESOIL contains all necessary parameters for 24 common industrial and agricultural pollutants.

Internal Data Transfer and Output Export become available

The data transfer between the WHI UnSat Suite models, as well as export model data for use with other applications or processing with spreadsheet programs (e.g. MS EXCEL) become available.

The WHI UnSat Suite Concept

The WHI UnSat Suite utilizes a unique interface design that accommodates the specialized features and analysis capabilities of each model, while maintaining a consistent look and feel for each data set. This allows you to easily switch from one model to another without having to go through the painful steps of learning a different interface and data structure for each new model.

The familiar design and functionality of the interface is extremely easy to learn and facilitates quick and easy access for conveniently modifying any of the model input parameters. The many interactive graphical tools allow you to quickly and easily create a model profile, modify the layer design, and customize the model properties and boundary conditions.

The project-oriented data management system allows you to easily manage multiple models, and different model types, within the same project database file. This project-oriented approach lets you conveniently navigate between different models and easily compare model input data and simulation results. In addition, the WHI UnSat Suite supports a teamwork environment where project files can be loaded, modified and run over a networked system.

There are seven key benefits that make UnSat Suite 2.1 the software package of choice for simulating unsaturated zone processes:

[1] Organizing Information is Easy

Information is organized into projects. Each project is associated with a name, an author, a client, and a location. This simple method of distinguishing projects makes project management easier.

[2] Specifying Site Conditions is Intuitive

All specific site data are organized in profiles which represent a unique combination of unsaturated zone structure and properties. Graphical tools available in the Project Tree and Project Profile View make visualizing and modifying of profiles simple.

[3] Describing a Problem is Easy

Soil and chemical properties are easy to edit and manage. The interface tools allow you to easily specify initial and time-varying boundary conditions of the site.

[4] Obtaining Accurate Weather Data is Simple

You can easily create a database of accurate weather information for your area of concern. WHI UnSat Suite allows you to select from over 3000 weather stations around the world to create statistically reliable weather data for practically anywhere on the globe.

[5] Model Output is Comprehensive

UnSat Suite 2.1 output allows you to view information in a broad, qualitative manner, as well as a detailed, quantitative form. You can quickly see the effects of different properties, designs, or situations as you adjust the parameters of your project.

[6] Presentation of Results is Easy

You can use the **Report Generator** to easily create and print your own report, which includes input information and selected output graphs and tables.

[7] Output Results May be Used by another Model or Exported

You may easily transfer data between the WHI UnSat Suite models, as well as export model data for use with other applications (e.g. export groundwater recharge assessed by Visual HELP to Visual MODFLOW).

[8] File Management is Flexible

Files are grouped in project sets and can be stored as well as edited or viewed by other users through a network connection.

How to Contact WHI

If, after reading this manual and using your software, you would like to contact Waterloo Hydrogeologic Inc. with comments, suggestions, or if you need technical assistance with your software installation, you can reach us at:

Waterloo Hydrogeologic Inc.

460 Phillip Street - Suite 101

Waterloo, Ontario, CANADA, N2L 5J2

Phone +1 (519) 746 1798

Fax +1 (519) 885 5262

Email: techsupport@waterloohydrogeologic.com

Web: www.waterloohydrogeologic.com

Software Maintenance and Technical Support

Waterloo Hydrogeologic's Annual Maintenance Contracts are designed to reduce the amount of time and effort required to keep your modeling software up-to-date. With automatic reminders of updates and upgrades, users will never have to wonder if their software is out-dated. Additionally, with unlimited technical support available during the term of the Maintenance Contract, software users can obtain rapid resolution to all of their software issues.

Annual Maintenance Benefits

- Free Major Software upgrades, and version updates
- Unlimited Telephone Support
- Unlimited email Support
- Priority Response to Error Reports
- Direct Bug Fix Updates
- Documentation Updates
- On-Demand FTP Downloads
- Showcase Your Projects in WHI E-News
- Option to Beta Test New Products
- Annual Reminder for Contract Renewal

For more information about Maintenance Contracts, please contact our Sales department via phone at +1 (519) 746-1798, or via email at Sales@waterloohydrogeologic.com

Other Products by WHI

Visual MODFLOW

A pre- and post processor for MODFLOW, MODPATH, and MT3D. A complete package for the visualization of model input and simulation results. The largest time-saving breakthrough for rigorous three-dimensional groundwater modeling since the release of MODFLOW.

FLOWPATH II

The most complete two dimensional, steady-state, groundwater flow and pathline model. It computes hydraulic heads, pathlines, travel times, velocities and water balances (verified against the USGS MODFLOW, approved by the U.S. EPA, and recommended by the IGWMC).

FLOWPATH II has been radically improved from its predecessor to include many new features and an enhanced graphical display to give you more power, flexibility and control than ever before. Also, FLOWPATH II now includes contaminant transport simulation capabilities.

FLONET/TRANS

A powerful yet easy-to-use two dimensional, steady-state groundwater flow and transient contaminant transport model. Calculates and displays

equipotential distribution, streamlines, flow nets, velocity vectors, and temporal graphs of concentration at multiple observation points.

AIRFLOW/SVE

The only comprehensive soil-vapour extraction model to simulate the coupled process of soil-vapour flow and multi-component vapour transport in the unsaturated zone.

PRINCE

A compilation of the ten Princeton Analytical Models which includes seven mass transport models (one-, two-, and three-dimensional) and 3 two-dimensional flow models.

AquiferTest

An easy-to-use graphically oriented package for estimating transmissivity, hydraulic conductivity and storage properties for a variety of aquifer types. The program contains analytical solutions for pumping tests and slug tests for confined, unconfined, and leaky confined aquifers.

Visual Groundwater

The first software package to combine state-of-the-art graphical technology for 3-D visualization and animation capabilities with an easy-to-use graphical interface designed specifically for environmental project applications.

Waterloo Hydrogeologic Inc. Training and Consulting

Waterloo Hydrogeologic Inc. offers individual tailored training courses on groundwater modeling. Our modeling courses emphasize on how to set up a proper groundwater model (grid design, boundary conditions, etc.), the interpretation of results (calibration, prediction, etc.), and extensive coverage on the mechanics of using groundwater models. Courses can be arranged by contacting Waterloo Hydrogeologic Inc.

Waterloo Hydrogeologic Inc. also offers expert consulting and peer review services for all numerical modeling problems concerning groundwater flow and mass transport. For further information contact Waterloo Hydrogeologic, Inc.

Acknowledgments

The development of UnSat Suite was directed by Dr. Mikhail Gogolev who designed the interface, tested the product and wrote the manual. Dr. Dmitri Eidenzon co-designed the interface and was responsible for

programming. Konstantin Choumak took responsibility for programming of version 2.2. Other members of the development team include Igor Evsikov, who developed the interface for the Weather Generator, Alexander Sapozhnikov, who converted the original FORTRAN codes to DLLs, Karin Nova, who developed sections of the database, and coop students Diane Cameron, Livia De Vellis and Bruce Davison, who prepared the new WGEN world database and took part in testing the interface and preparing the manual and on-line help. Dr. Serguei Shmakov, Douglas Graham, Gary Moore and Paul Martin provided support, as well as Dr. Nilson Guiguer and Patrick Delaney, who are responsible for overall software development at Waterloo Hydrogeologic, Inc.

The software research and development was supported financially by the National Research Council of Canada through the Industrial Research Adaptation Program (#28557U), which is greatly appreciated. In particular, the development team wishes to thank the Industrial Technology Advisor, Dr. Ernie Davison for his support.

1

Installation and Start-Up

System Requirements

To run WHI UnSat Suite, you will need the following **minimum** system configuration:

- Pentium II based computer;
- 32 MB RAM (128 MB recommended);
- CD-ROM drive for software installation;
- A hard drive with at least 60 Mb of free space;
- Windows 98/Windows NT (SP4)/Windows 2000/Windows XP.

The following fonts should be installed on your computer: MS Sans Serif, Arial, Times New Roman, and Courier New.

Installing WHI UnSat Suite 2.2 from the CD-Rom

A PDF file of the WHI UnSat Suite Users Manual, and extracts from the original tutorial guides for the WHI UnSat Suite models, are included on the Installation CD ROM in the Manual subfolder. If you do not already have Acrobat Reader, simply select the **Get Acrobat Reader** link found in the opening screen.

Please follow the steps below to install your software from the CD-ROM.

- [1] If applicable, log in as the Local Administrator of your computer, or as a user with Local Power User rights.
- [2] Insert the program CD into your CD-ROM drive to initiate the CD Navigator. If the navigator does not automatically open the Product Installation screen, explore the CD-ROM drive and double click on the **Install32.exe** file.

- [3] To begin installing your software to your hard disk, click on “**WHI UnSat Suite Plus Installation**”
- [4] The **Welcome** window will appear. Click [**Next>**] to continue, and follow the Installation Wizard to enter your product Serial Number (please remember to use CAPITAL letters), select your installation directory, and select the components to install.

Starting WHI UnSat Suite

After installing WHI UnSat Suite, click **Start** on the Windows Taskbar, choose **Programs**, go to the program file where WHI UnSat Suite is located, and click **UnSat22** to open the product.



The installation program will also place an icon on your desktop, which you can also click to start WHI UnSat Suite.

2

General Features

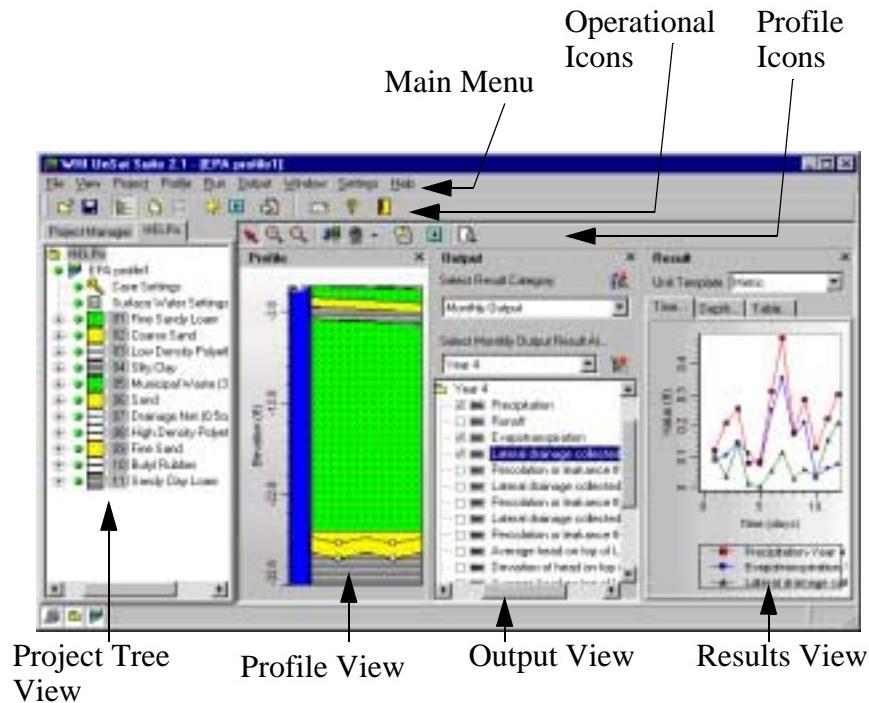
Terms and Notation

The following terms and notations will be used throughout the manual:

type	Type in the given word or value
select	Click the left mouse button where indicated
↵	Press the <Enter> key
☞	Click the left mouse button where indicated
☞ ☞	Double-click the left mouse button where indicated
[...]	denotes a button to click.
...\\...\\...	denotes a menu selection.

Getting Around the Interface

Below is an example of the WHI UnSat Suite windows interface using the HELP model:



After starting and loading WHI UnSat Suite, a window will appear on your screen. The window will be blank except for the project tree view, as you have not chosen a problem or a model, or generated any output. The previous window illustrates all of the display features of UnSat Suite available after a model has been run. The Main Menu items are at the top, Operational Icons are just below, and the Profile Icons are just above the Profile View. The Project Tree View is to the left of the main window, the Profile View is in the middle, and the chart and Table Output Views are on the right. The menu items, icons, and views are described below.

Main Menu

The top menu bar contains the following menu options:

File	Create a project, open/close/save/delete a project, or exit UnSat Suite. You may also open previously saved sets of projects, archive current projects and run projects stored on other machines through the local area network.
View	View/close Project Tree, Profile View, Output Structure, and Results View.
Project	View project properties, create a new profile, or delete a profile.

Profile	View profile properties, view layer group geometry for HELP, view/edit stress periods for VS2DT.
Run	Run the selected model or run the Weather Generator (HELP) or view and edit stress period parameters (VS2DT).
Output	View simulation results and input file or clear the Results View.
Window	Arrange the windows that present information about different profiles.
Settings	Create, edit, or delete profile templates, materials, authors, clients, locations, and unit templates. You can also restore corrupted project clicking the Repair button.
Help	Go to the Help window and general information on UnSat Suite.

Operational Icons

The operational icons provide you with quick access to functions related to projects.



Create New Project Click this icon to begin a new project.



Save Project

Click this icon to save a project with its current settings.



Open/Close the Project Tree View Click this icon to open and close the Project Tree View if you need more space for the graphs.



Create New Profile

Choose this icon to create a new profile in the open project.



View Profile

Choose this icon to view the profile that is highlighted in the Project Tree View.



Run Weather Generator (HELP only)

Choose this icon to run the Weather Generator.



Run the current model for all profiles

Choose this icon to run the model simulation for all profiles in the open project.

	Prepare a Report	Choose this icon to prepare a report after you have run the simulation
	Remote Data Access	Choose this icon to open the archived project set or to open the project stored on the other machine through the LAN. This icon is visible only when all projects are closed.
	Copy Holder	Use this icon to temporary store your projects while you are switching from one project set to another. To save a copy of the project in the Copy Holder simply drag and drop the project from the Project Tree. This icon is visible only when all projects are closed
	Mail To...	Click this icon to mail to the WHI's technical support group.
	Help Topics	Click this icon for on-line help.
	Exit	Click this icon to exit UnSat Suite.

Profile Icons

The profile icons provide you with quick access to functions related to profiles.

	View...	Click this icon to switch from zooming mode to view mode.
	Zoom In	Click this icon to zoom in on the profile.
	Zoom Out	Click this icon to zoom the profile out.
	Delete Layer(s)	Choose this icon to delete selected layers in the profile view (HELP).
	Restore...	Choose this icon to restore any layers that have been deleted since the last time you saved the project (HELP).



Profile Properties Choose this icon to view the profile properties.



Run Model For Profile

Choose this icon to run the HELP model for the active profile in the open project.



Print Preview

Click this icon to preview and print the profile picture.

Project Tree View

Organize your projects and profiles in the Project Tree View.

Profile View

View the profile cross-section in the profile view.

Output View

Output is organized into labeled information folders.

Results View

The Results View has three tabs. Each tab presents the data in a different format.

Time... The **Time...** tab shows a plot of the selected output.

Depth... The **Depth...** tab shows a plot of the selected output vs. the depth of the profile.

Table... The **Table...** tab shows a table of the selected output.

Starting a New Project

WHI UnSat Suite makes it easy for you to organize your activities into projects. Projects are identified by a unique project name, and are associated with:

- a problem and a model,
- an author (or a person in charge of the project),
- a client,

- and a project location (for the HELP model).

To create a new project:

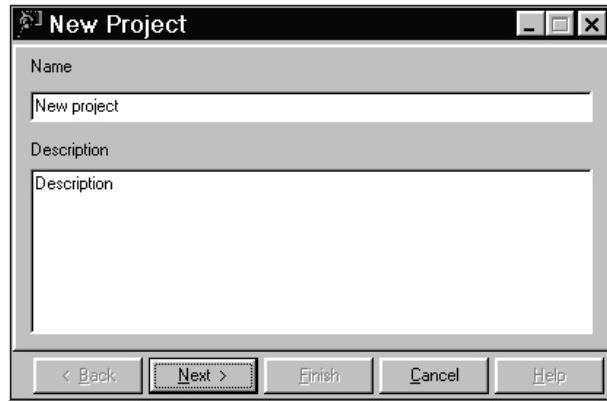
From the **File** menu,  **New Project**.

OR



 on the toolbar.

The **New Project** dialog box will appear:



The new project wizard will guide you through the steps required to begin a new project.

General Project Information

Click in the **Name** box, and type a concise name for your project (maximum 35 characters).

In the **Description** box, you can type comments about your project. If you type a full name for your project here, you will be able to use it on the title page of your UnSat Suite report.

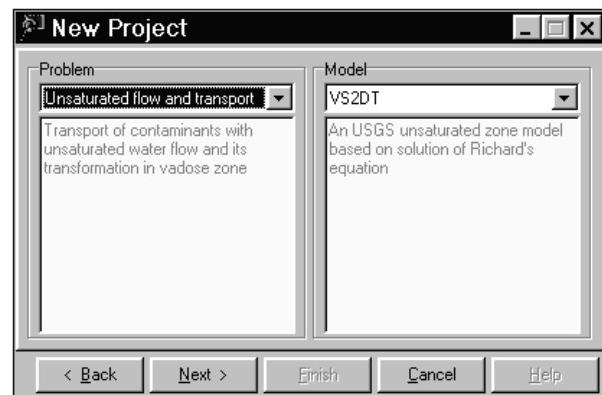
 [Next>]

Selecting a Problem and Model

Using the chart below determine the model which is most appropriate for your scenario:

MODEL	PROBLEM	MODEL DESCRIPTION
HELP	Landfill Hydrology	A versatile US EPA model for predicting landfill hydrologic processes and testing of effectiveness of landfill designs.
PESTAN	Pesticide Contamination	A popular US EPA model for making assessments of contamination of soil and groundwater with pesticides.
VS2DT	Unsaturated Flow and Transport	A US Geological Survey model for describing the transport of contaminants with unsaturated water flow and its transformation in vadose zone.
VLEACH	Hydrocarbons and VOUCH's	A popular US EPA model for making assessments of contamination of soil and groundwater with volatile organic contaminants.
SESOIL	Seasonal Flow and Transport	A popular US EPA model for long-term simulations of chemical transport and transformations in soil.

From the **Problem** list choose the appropriate problem.



The **Model** list will automatically display the correct model to use.

[Next>]

Location

If you selected the HELP or SESOIL models, the wizard will prompt you for the location of your site:



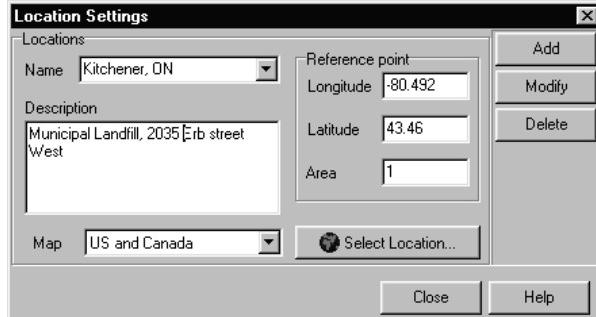
The location determines the amount of solar radiation at the project site.

If you did not select HELP or SESOIL, this section will automatically be omitted.



Select the location of your project from the **Location** list, or to choose a location that is not on the list, click [**Locations...**].

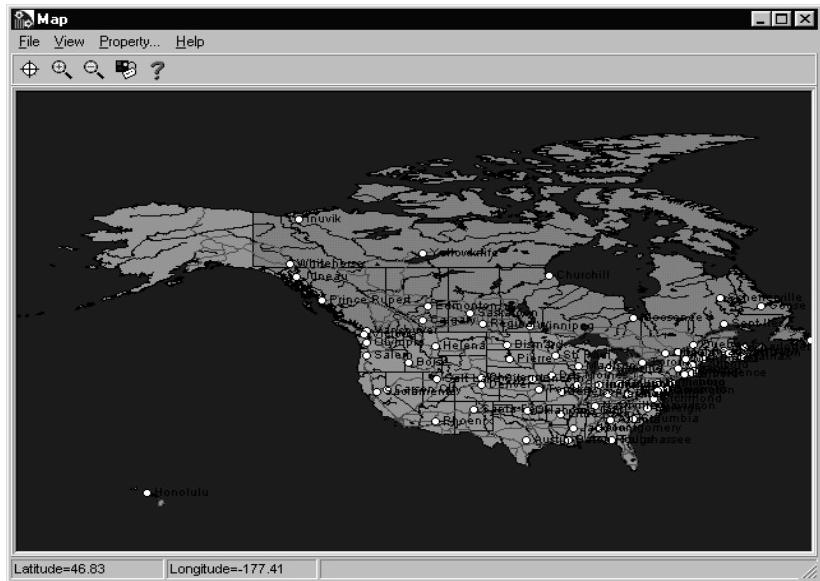
The **Location Settings** dialog box will appear:



Type the name and a description of your problem in the appropriate boxes. Also, type the surface area in the **Area** box. The area will be measured in hectares if you are using the metric unit system, and acres if you chose the customary unit system.

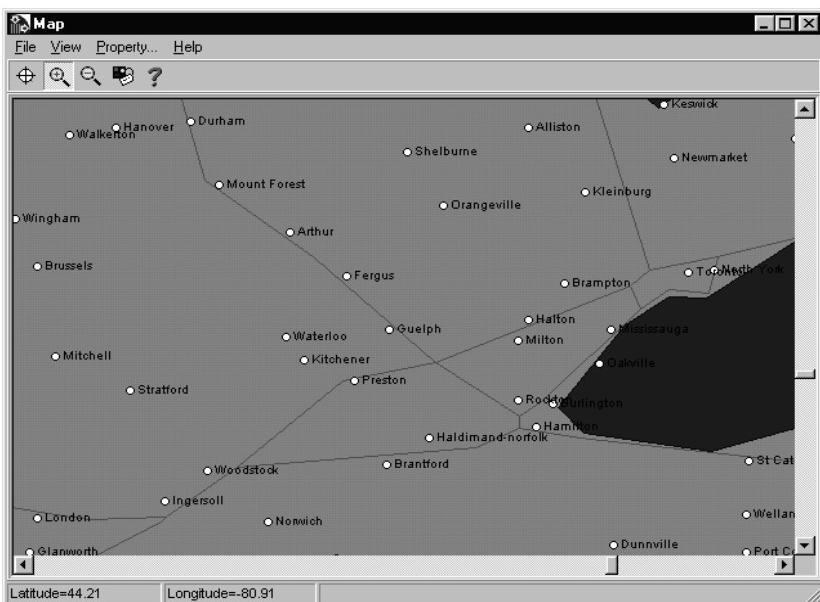


You can type the co-ordinates of your site in the **Longitude** and **Latitude** boxes, or click [**Select Location...**] to use the GIS Map. Choose a GIS map from the list. There are 8 maps available. These maps include Africa, Asia, Australia, Europe, North America, South America, the Former USSR, and the US.



Click and drag a zoom rectangle around your site.

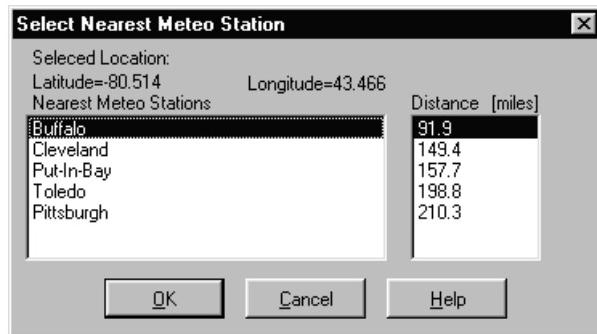
Repeat until you see cities near your site.



to activate the cross-hairs.

Click your site with the crosshairs.

The **Location Settings** dialog box will reappear:



Select one of the stations that best represents your climate conditions and click **OK**.

You will return back to the **Location Settings** dialog box. Notice that the longitude and latitude have been updated to your site.

Note: For this GIS searcher, a longitudinal value that is negative indicates west, while a positive value indicates east relative to the prime meridian. Similarly, a negative value for latitude is south, while a positive value is north relative to the equator.

☞ **[Add]** to save the current location to the locations database. You may edit the location name first.

☞ **[Modify]** to save the changes to the current location.

☞ **[Delete]** to delete the current location from the list.

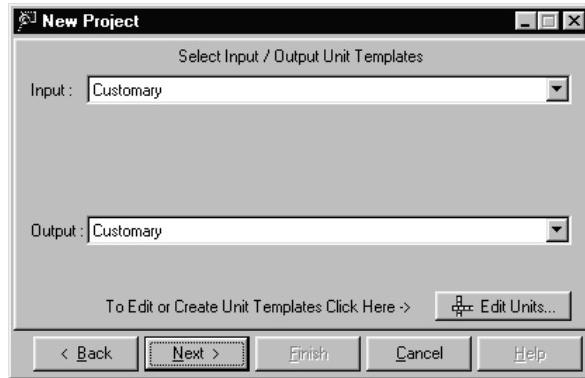
☞ **[Close]** to return to the **New Project** dialog box.

Back in the **New Project** dialog box:

☞ the locations drop-down list. Your recent selected location will be placed in the list in alphabetical order. Click it to make it active for the project.

☞ **[Next>]**

Units



UnSat Suite allows you to specify the units that you will use for your input, and the units for your results. The input and output unit templates can differ which provides the user with more flexibility (e.g. you may input layer thicknesses, pressure heads and other parameters which have units of length in inches and view the results in meters).

You can also change units for all input parameters throughout the design process.

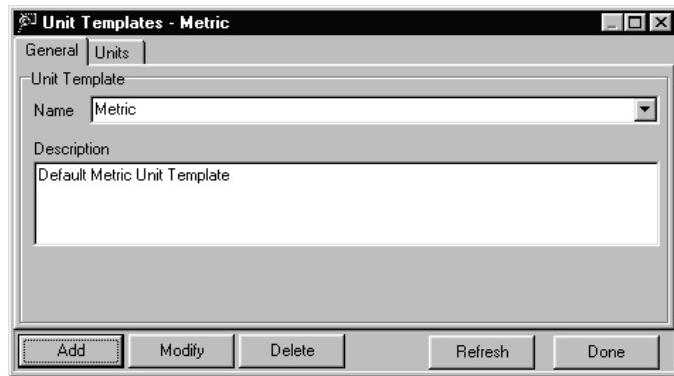
To change the units for any input parameter:

- 1) **Unit** field beside the **Value** for the appropriate parameter. The drop-down list will show all units available for the parameter.
- 2) **the convenient unit.**
- 3) input the parameter value in this unit without any conversions
- 4) **OK**

UnSat Suite will convert the value into the units used by the model automatically.

Select the input unit system from the **Select Input/Output Unit Templates** drop-down list. Select the output units from the **Select Output Unit System** drop-down list. If you are using the HELP model, you may select the unit system for the HELP's original listing. You are limited to Metric and Customary units here.

To view the **Unit Templates** dialog box, click [**Units**].



Click the drop-down arrow in the right part of the **Name** text box to view the list of available **Unit Templates**.

Select the desired template from the list and click the **Units** tab.

The **Unit Template** dialog box will appear for selected template.

Here you can change units to use in your projects		
Unit Type	Unit	Type Description
Profile Mass Balance	g/m ²	The mass of chemical in profile per unit area
Bulk Density	g/cm ³	The mass of dry soil relative to the bulk volume of soil
Area	ha	Area characterized by specific profile
Slope	‰	Slope
Defect Density	#/ha	# of defects per unit area in result of geomembrane installation
Transmissivity	cm ² /sec	the product of saturated hydraulic conductivity and thickness c
Moisture Content	vol/vol	moisture content of a unit soil volume, specific values of full sat
Dispersion Coefficient	cm ² /day	Units for coefficients of hydrodynamic dispersion and molecule
Length	m	Length
Sorption Constant	ml/g	The coefficient of distribution of the pollutant between the solid
Degradation Constant	/hr	The coefficient used in the first order decay or degradation ma
Concentration	mg/l	The amount of chemical per unit volume of solution
None	-	No units at all.
Specific Storage	1/cm	Change in water content under the unit pressure change due to
Time	days	Time
Mass	mg	A measure of the amount of matter
Root Activity	1/cm ²	Root Activity
Mass Flux	g/day	The mass of chemical entering the soil or passing the node in .
Surface Resistance	1/m	Surface resistance for evaporation
Fraction	‰	Ratio of the part to the whole.
Ion-exchange Capacity	g-equiv/g	The amount of exchangeable cations per unit mass of soil
Flow Velocity	cm/day	Flux, Recharge Rate, Hydraulic Conductivity, Subsurface Inflow
Concentration in Soil	g/kg	The amount of chemical per unit mass of soil

In unit templates, all units used in UnSat Suite are grouped into **Unit Types**. One unit type may include units for several types of parameters. For instance, parameters Flux (VS2DT), Recharge Rate (PESTAN, LEACH), Hydraulic Conductivity (all models), and Subsurface Inflow (HELP) all use the same units: length divided by time. All these parameter's units are grouped into the category **Flow Velocity**.

To view available units for specific unit type, click in the **Unit** field next to the **Unit Type**. The drop-down list will appear. Below you can see available units for the **Flow Velocity** unit type.



To create a new Unit Template:

- 1) **Units...** to open an existing template
- or from the **File** menu click **Settings** and then **Units**
- 2) Type a unique name in the **Name** box.
- 3) Type a description in the **Description** box.
- 4)
- 5) To edit units, click the units from the appropriate list.
- 6) [Add]

Now you can select the newly added unit template for the input or output.

The output template can also be changed after the project is complete if you want to view the output in different units.

To view the output in different units after the project is set:

- 1) From the **Project** menu click **Properties**
- 2)
- 3) select the appropriate output template from the list
- 4)
- 6) run the model again

You can modify a template's default units. To modify a template, select the template from the list. Then, make the necessary changes.

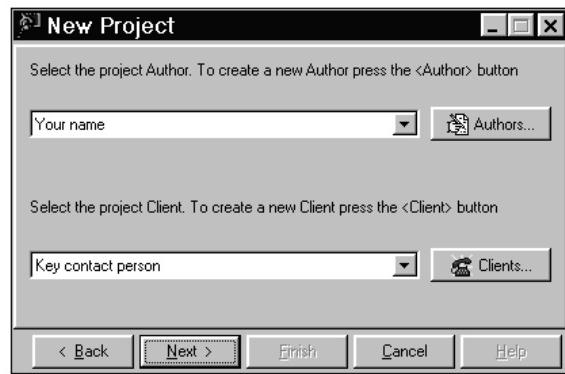
☞ [Modify] to save the changes.

To delete a template, select the template from the list. Then, ☞ [Delete] to delete the template. However, you are not able to delete default Metric and Customary templates.

☞  to exit Unit Templates.

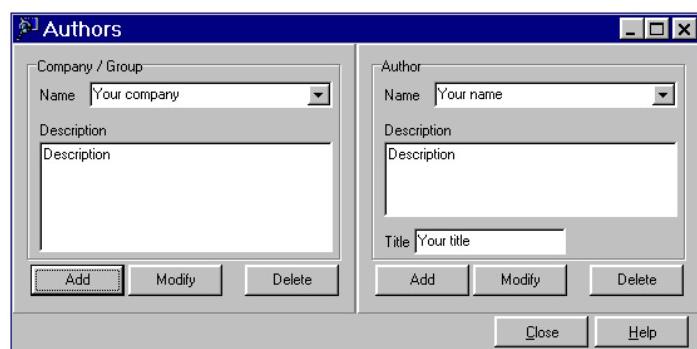
☞ [Next>]

Authors and Clients



Click an existing author from the list, or create a new author by clicking [Authors].

The Authors dialog box will appear:



Each author is associated with a company.

Company/Group

In this area, you can add, modify, or delete companies and groups. To add a new company, edit the information in the **Name** box and the

Description box, and click [**Add**]. To update an old company with current information, edit the name and description, and click [**Modify**]. To delete a company, select the company or group from the list, and click [**Delete**].

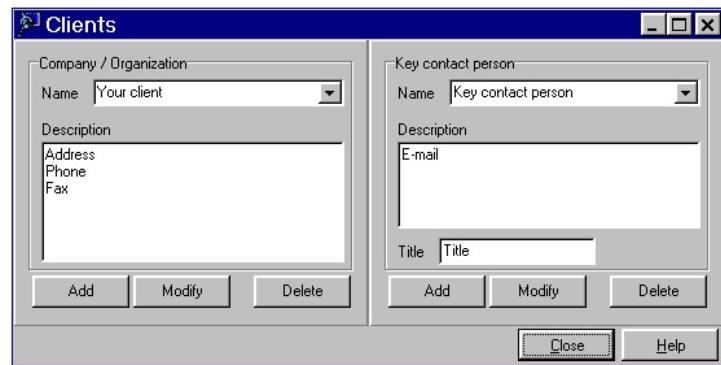
Author

In this area, you can add, modify, or delete authors from companies. To add a new author, edit the information in the **Name** box, **Title** box, and **Description** box, and click [**Add**]. To update the information about an author, edit the information, and click [**Modify**]. To delete an author, select the author from the list, and click [**Delete**].

☞[**Close**] to return to the **New Project** dialog box.

Click an existing client from the list, or create a new client by clicking [**Clients**].

The **Clients** dialog box will appear



Each contact is associated with a company or organization.

Company/Organization

In this section you can add, modify, or delete companies and organizations. To add a new company, edit the information in the **Name** box and **Description** box, and click [**Add**]. To update an old company with current information, edit the information, and click [**Modify**]. To delete a company, select the company from the list, and click [**Delete**].

Key Contact Person

In this area you can add, modify, or delete contacts from companies. To add a new contact, edit the information in the **Name** box, **Title** box, and **Description** box, and click [**Add**]. To update the information about a client, edit the information, and click [**Modify**]. To delete a client, select the client from the list, and click [**Delete**].

☞[**Close**] to return to the **New Project** dialog box.

☞[**Next>**]

The final step to create a new project is determining if the parameters are correct.



If the information presented is correct **[Finish]**. You can go back and edit any step in the wizard by **[<Back]**. You can cancel the new project at any time by **[Cancel]**.

After the project is started, the **New Profile Wizard** dialog box will open. The wizard will take you step by step through the process of selecting a new profile. For complete instructions on choosing an existing template or creating a new template, refer to the later sections of this chapter.

Loading an Existing Project

All existing projects are displayed in the Project Tree. To open a project, double click on the project in the Project Tree.

OR

To open the project, <right click> on the project in the Project Tree. Click **Open**.

OR

To open the project, highlight the project and from the **File** menu click **Open Project**.

Profiles in UnSat Suite

All information about the natural conditions of the project site, unsaturated zone layering, soil parameters and parameters of the engineering systems installed in profile, as well as information about the contaminant loads is stored in the **UnSat Suite profiles**. The UnSat Suite models perform simulations only for information arranged in profile sets. Consequently, profile is a main category of data arrangement in UnSat

Suite. The profile is associated with the area which it represents. Actual project sites will rarely be uniform. In this case the whole area of the site has to be split into a number of sub-areas with unique profile construction, material parameters and boundary conditions.

Two different types of profiles exist in WHI UnSat Suite which originate from the models included in the package.

Designing profile. The first type of profile is a **designing** type. This type of profile is simulated with the HELP model. Civil engineers use this model to try different landfill designs and try the optimal combination of the landfill performance and cost. The thickness of individual layers and the total profile depth are the terms of the optimization equations and can be edited by the user at any time.

Using the HELP model, WHI UnSat Suite allows the user to fix the top elevation or the bottom elevation of the designing profile. The top elevation can be fixed when there is a constraint on the appearance of the landfill. The bottom elevation can be fixed when there is a constraint on the depth of the landfill (e.g. the bottom of the landfill should be above the highest groundwater level). When the user edits the thickness of a layer, the profile will change differently, depending on whether the top or bottom elevation was fixed in the Profile Properties dialog box. If the profile top was fixed and the user increased the thickness of a layer, the profile will grow downwards. If the profile bottom was fixed and the user increased the thickness of a layer, the profile will grow upwards.

Natural Profile. VS2DT is a model for simulation of pollutant flow, transport and transformation within the natural profiles of the vadose zone. Working with these profiles, the user usually knows only the depth of the unsaturated zone, while the exact layer structure is commonly unknown.

Applying VS2DT, the user can set the top and bottom elevation of the profile and edit them at any time during the input data preparation. However, editing of the layer thickness by correcting the thickness value is not allowed. To manipulate the soil layer structure within the fixed profile depth, the user has a set of graphical tools with which he/she can move the layer boundary, split and merge layers, and change the soil type within the layer. When the user moves the upper profile boundary graphically or changes the top profile elevation, the introduced changes will impact only the thickness of the upper layer. Similarly, a move of the upper profile boundary or changes to the bottom profile elevation, will change only the thickness of the lower layer.

PESTAN and VLEACH are also models which simulate the natural profiles. However, they can simulate only a single-layer profile.

Creating a New Profile Using a Profile Template

To use the profile template for creating of a new profile, you must have a project loaded. Once the project is loaded, click **New Profile** from the **Project** menu. You may also click the following icon from the Operational Toolbar:



The **New Profile Wizard** dialog box will open.



☞ [Next>].

Choose the template you wish to open from the **Available templates** list.

☞ [Next>].

If the information presented is correct ☞ [Finish]. You can go back and edit any step in the wizard by ☞ [<Back]. You can cancel the new profile at any time by ☞ [Cancel].

Creating a New Profile

To create a new profile, you must have a project loaded. Once the project is loaded, click **New Profile** from the **Project** menu. You may also click the following icon from the Operational Toolbar:



The **New Profile Wizard** dialog box will open.



Click **create new profile**.

☞ [Next>].



Type in the values for the elevation parameters: **Top**, and **Bottom**. **Thickness** will be the difference between **Top** and **Bottom**.

Select a material category from the **Material Category** list. Then, select a material from the **Material** list. The choices for materials and categories will vary depending on the model.

Note: The New Profile Wizard only allows you to input one layer. Additional layers for VS2DT and HELP can be added once the project is open.

☞ [Next>].

If the information presented is correct ☞ [Finish]. You can go back and edit any step in the wizard by ☞ [<Back]. You can cancel the new profile at any time by ☞ [Cancel].

From the **File** menu, ☞ **Save Project**.

Creating a New Profile Template

In addition to using one of the distribution templates, you can create your own template to use in your projects.

Create a new profile using an existing template or create a new template, and edit it until you are satisfied that it will suit your needs.

To create a new template:

- 1) From the **File** menu, **Save Project**. (this is necessary to save the template)
- 2) <right click> on the profile in the Project Tree
- 3) **Save As Template**
- 4) **[Yes]**
- 5) Type a unique name for this profile template in the **Confirm Name** dialog box.
- 6) **[OK]** to save a template.

Starting the Simulation

To run the simulation, from the **Run** menu click the name of the model.

Once the simulation is complete, you can view the output.

Using the Material Designer

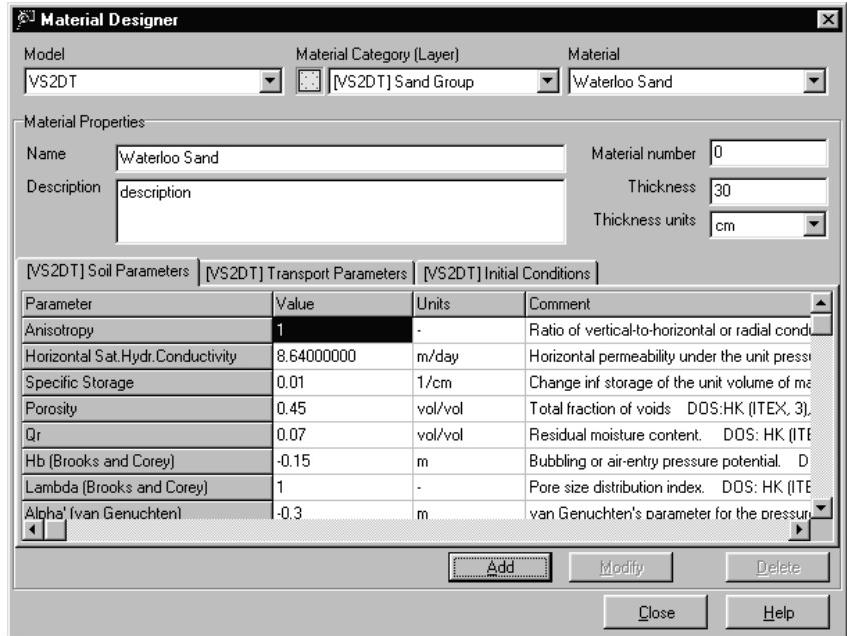
The **Material Designer** lets the user maintain and edit the material database of UnSat Suite. Both the materials comprising the profile and the pesticides/contaminants leaching through the soil are stored in the material database.

You can create new materials for each model with the Material Designer. To begin, open the dialog box.

To open the dialog box:

- 1) From the **File** menu click **Settings**
- 2) click **Material Designer**.

Materials are stored separately for every model. To use the **Material Designer**, select the name of the model from the **Model** list. Then, select the layer type from the **Material Category** list. Finally, select the material from the **Material** list.



For VS2DT, PESTAN and VLEACH in the Material Designer all soils are structured according to the widely used USDA-SCS classification scheme into the material groups (categories). For the HELP model all materials are grouped according to the functional categories of landfill materials. Within each category the user may add an unlimited number of soil records.

The following chart illustrates the components of the 12 main material categories for the VS2DT, SESOIL, PESTAN and VLEACH models.

yellow color 	Sand	Between 85-100% sand, under 10% clay, under 15% silt.
yellow color 	Loamy Sand	Between 70-90% sand, under 15% clay, under 30% silt.
turquoise color 	Sandy Loam	Between 42-85% sand, under 20% clay, under 50% silt.
turquoise color 	Sandy Clay Loam	Between 45-80% sand, between 20-35% clay, under 27% silt.

green color 	Silt	Under 20% sand, under 10% clay, between 80-100% silt.
turquoise color 	Silt Loam	Under 50% sand, under 28% clay, between 50-90% silt.
turquoise color 	Clay Loam	Between 20-45% sand, between 28-40% clay, between 15-52% silt.
turquoise color 	Loam	Between 22-53% sand, between 9-28% clay, between 27-50% silt.
turquoise color 	Silty Clay Loam	Under 20% sand, between 28-40% clay, between 40-72% silt.
grey color 	Sandy Clay	Between 45-65% sand, between 35-55% clay, under 20% silt.
grey color 	Silty Clay	Under 20% sand, between 40-60% clay, between 40-60% silt.
grey color 	Clay	Under 45% sand, between 40-100% clay, under 40% silt.

The soils were composed of silt ($2\text{-}50 \mu\text{m}$), clay ($<2 \mu\text{m}$), and sand ($>50 \mu\text{m}$).

The view of the Material Designer dialog box differs for the HELP model. The **Design Layer Permissions** frame, which does not show up for the rest of the models, displays features which pertain to specific HELP material categories.

The **custom thickness** refers to the thickness of the layer and can be edited directly in the **Material Designer** or can be edited through **Layer Properties**. Manufactured materials, such as geomembrane liners, have set thicknesses which can not be altered. The interface allows setting **Drainage** parameters for **Lateral Drainage** and **Geotextiles and Geonets** layers in HELP. The specific drainage parameters of the layer can be specified in layer properties while running the project. Also in HELP, the user can specify the initial moisture content of some layer categories. If **custom initial moisture content** for the layer is checked, the user can specify this parameter through the layer properties.

To add new material to the **Material Designer**, use one of the existing material records. In the **Material Properties** frame, change the existing

name to a unique name for the new material in the **Name** box. Then, enter a description of your new material in the **Description** box.

In the **Material number** box you can enter an index number. For the HELP model, material numbers 1 - 42 are used by the default materials. New materials for the HELP model should begin at 43. For the remaining models, numbering can begin at one. This number is for filing purposes only and does not change the properties of the material. It is important, however, to make sure that the model numbers do not repeat. Each new material, within a model, should have a new number.

In the **Thickness** box you can enter the default thickness of the material. Every time you enter the material into a profile, it will be entered at this thickness. Click the accompanying units for the thickness in the **Thickness units** box.

The **Soil Parameters** tab will display all the editable properties of the soil. You can specify the materials default property values in the appropriate boxes in the **Value** column. You can also click the accompanying units from the **Units** column.

If you are designing or modeling for the VS2DT model, there will be additional tabs for **Transport Parameters** and **Initial Conditions**. To edit these parameters click the appropriate tab. Type in the new value in the appropriate boxes in the **Value** column. You can click and select other units from the **Units** column if you have your data in different units.

When you have completed specifying the properties of the new material,  [Add].

Adding new contaminant to the database

In addition to adding new soil materials to the database, you can add new contaminants. Pesticide and contaminant categories present in each model are listed in the **Material Category** list along with the various soil categories. New pesticides/contaminants can be added using the same procedure outlined above.

Modifying and Deleting Materials

You can modify a material's default properties. To modify a material, select the material from the appropriate lists. Then, make the necessary changes to the material's properties.  [Modify] to save the changes.

To delete a material, select the material from the appropriate lists. Then,  [Delete] to delete the material.

To close the **Material Designer** dialog box,  [Close].

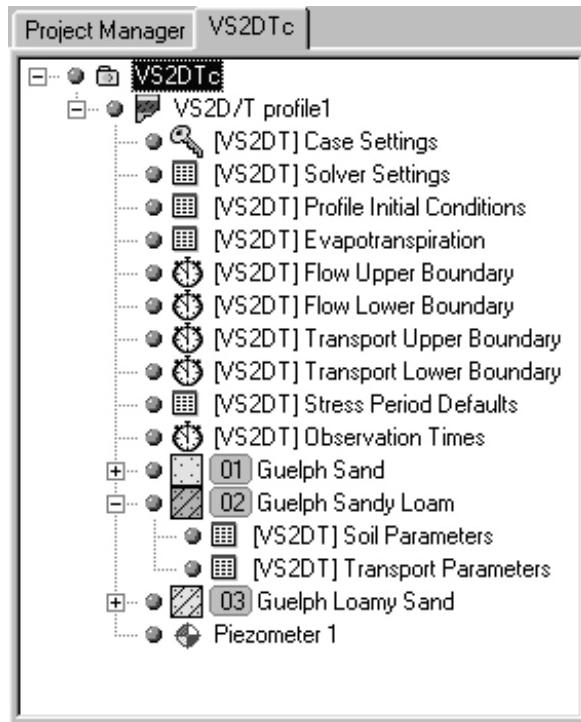
3

Common Tools

Working in the Project Tree View

Elements of the Profile Structure

Although different models require different data, all projects in UnSat Suite have common structures to input and edit data. The following picture shows these structures using the default VS2DT profile as an example.



The Project Tree View shows the structure of the project named **VS2DTc** which contains one profile **VS2DT profile 1**.

The objects shown in the tree below the **VS2DT profile 1** are used to specify the profile settings. These objects are divided into two categories:

- pertaining to a profile
- pertaining to a specific layer within the profile

The objects starting from the [VS2DT] **Case Settings** to the [VS2DT] **Observation Times** are the profile parameter groups. They contain parameters pertaining to the whole profile.

The objects starting from the **Guelph Sand** to the **Guelph Loamy Sand** are profile layers. Layers have one or more layer parameter groups. In the picture the layer **Guelph Sandy Loam** is opened and the **Soil Parameters** and **Transport Parameters** layer parameter groups are shown.

Profile Parameter Groups

There are three categories of the profile parameter groups which perform different functions and are marked with different icons:

-  **Case Settings** - parameters from this group are used to set the major functions of the model and characteristics of the simulated case. Choices made by the user here may determine content of some other profile and layer parameter groups or even make them to appear or disappear.
-  **Regular Parameter Group** - this category of parameter groups is used to specify the values of single-value parameters. The value may not be a number, parameter may be a qualitative. In the last case the user has a list of qualitative values to choose from.
-  **Time Dependent Parameter Group** - this category of parameter groups is used to input time dependent parameters (e.g. types and values of boundary conditions, observation times).

Materials in Profile

Two types of materials can exist in a profile:

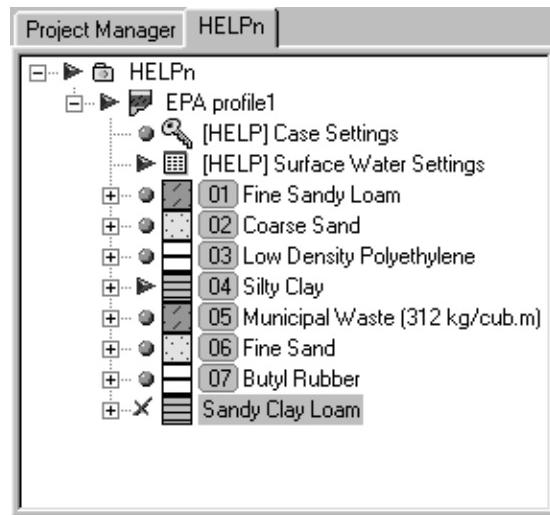
-  **Chemical** - in PESTAN and VLEACH this is a specific agricultural or industrial pollutant distributed within the profile and characterized with its specific transport parameters.
-  **Layer Material** - this can be a natural soil or rock (all models) or artificial materials in HELP (e.g. geomembranes, geonets). Layer materials has specified location within the profile and are characterized with top/bottom elevation, thickness and specific material parameters.

Observation Points

-  **Observation Point** - this is a depth at which a detail output is required. In future releases of UnSat Suite the observation points will have an interface to compare observed and simulated variables to calibrate the model.

Indicators of the Object's Status

UnSat Suite has symbols used throughout the program to denote changes in the profiles. The picture below illustrates these symbols:



In the above diagram, a parameter or parameters of the **Silty Clay** layer and the **Surface Water Settings** group were edited. The red triangles indicate the changes. The green dots are all the unaffected layers or groups. **EPA Profile 1**, and the project **HELP** are marked with the red triangle to show that they were somehow changed by the edit.

The **Sandy Clay Loam** layer was deleted. The delete is shown by a red 'x'. The red 'x' lets the user know that the object has been deleted, but can be restored. If the object is restored it will have a red triangle to show that it was altered.

An object cannot be restored once the project has been saved. After saving, the changes are permanent. The only way to go back to a previous project is to recreate it. To show that the project has been saved, the red triangles become green dots and the red 'x's with the names of the deleted objects disappear.

Accessing and Editing Objects of Profile Structure

Operations with the Objects of Profile Structure

Different operations are allowed for the different objects of the profile structure. While the parameter groups may be only edited, the chemicals may be edited or substituted with other chemical. The observation points can be edited or deleted.

All layer materials may be edited or substituted with other material. In addition, the layer structure in **HELP** and **VS2DT** can be altered to allow the user to create the specific profile structure. The way of altering profile structure depends on the model and type of profiles as it is described in Chapter 3.

Using the **HELP** model the user works with **designing profiles**. In **HELP** interface which is developed to make the landfill designing easy, the user may:

- resize any layer editing its thickness or graphically,
- edit parameters of the layer's slope,
- substitute material in the layer,
- delete the layer,
- split the layer,
- insert a layer above the current,
- resize the adjacent layers graphically by moving the layer's boundary,
- and insert a layer inside the current after using the split layer function.

The total profile thickness may change in result of layers editing. The elevation of profile will change depending on either the fixed top or the fixed bottom condition was chosen.

Using the **VS2DT** model the user works with **natural profiles**. In **VS2DT** interface which is developed to make multiple simulation of natural profiles easy, the user may:

- edit profile's top and bottom elevation,
- substitute material in the layer,
- split the layer,
- merge adjacent layers,
- resize the adjacent layers graphically by moving the layer's boundary.

The total profile thickness and elevation usually remain unchanged in result of layers editing. It will change only if the user changes top and bottom elevation. This action will cause resizing of the top or bottom layers.

Accessing Objects

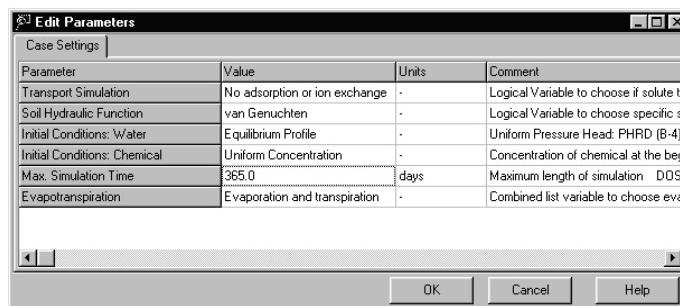
There are two general ways of accessing the objects of profile structure in UnSat Suite. You may either:

- double click the name of the object with the left mouse button
- or click the name of the object with the right mouse button and then click the desired function with the right mouse button

Customizing the Case Settings and a Regular Parameter Group

Case Settings group is used to set the major functions of the model and characteristics of the simulated case. Choices made by the user here may determine content of some other profile and layer parameter groups or even make them to appear or disappear.

The way to customize the case settings is shown using this group for the default VS2DT profile:



The **Case Settings** dialog box contains four fields: **Parameters**, **Value**, **Units** and **Comment**. From the six parameters shown, five have qualitative values and only **Max. Simulation Time** has a quantitative value. However, the value for all parameters can be edited or altered.

☞ the **Value** field to the right of the **Max. Simulation Time** to edit the value of this parameter. The cell will get highlighted as it is shown in the picture. If you click the left mouse button one more time the current value would get highlighted:



If now you type your value in the cell, it will replace the previous one. If, instead, you place the cursor at the certain position within the highlighted number and click again, the position of the cursor will become the insertion point from which you can start editing the value.

As mentioned previously, UnSat Suite allows you to change units at any time within the data preparation process. To change units for the **Max. Simulation Time**, click the **Units** cell beside the **Value** cell. The list of allowable units will appear:



From the list you may select the desired units by clicking them. Your previously entered value will be recalculated for the new units automatically.

To alter the value of the qualitative parameter, you have to double click in the **Value** field beside its name. The following list will open if you double click the **Value** cell for **Transport Simulation**:

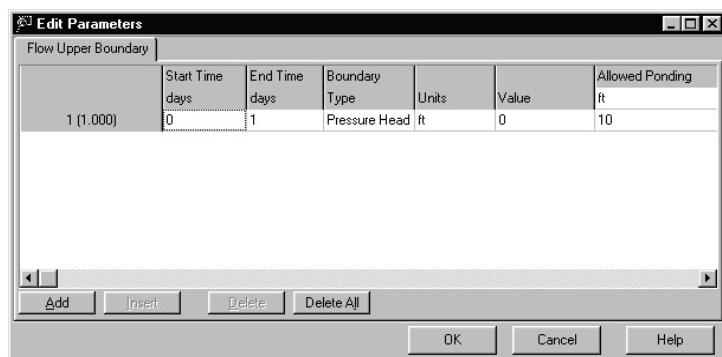


To select one of the options, click on it.

Regular Parameter Groups contain generally parameters with numeric values but the way of editing these values is the same as for the **Max. Simulation Time**.

Editing a Time Dependent Parameter Group

The way to edit parameters from this category of parameter groups will be shown on example of the **Flow Upper Boundary** group from the **VS2DT** interface. The first time you open this group, it will look the following way:



Let suggest that you have to input time dependent values for three periods 10 days long each and two periods 30 days long each.

To enter the length of the first period, click in the **End Time** cell and type **10** (the end time of the first period).

Now, to multiply the time period, click twice the **Add** button at the bottom of the dialog window. By default, the program will copy the current line of the table.

To change the period length, type value **60** in the **End Time** cell for the fourth period. Click the **Add** button to duplicate the 30 day period. The dialog window will look the following way:

	Start Time days	End Time days	Boundary Type	Units	Value	Allowed Ponding ft
1 (10.000)	0	10	Pressure Head	ft	0	10
2 (10.000)	10	20	Pressure Head	ft	0	10
3 (10.000)	20	30	Pressure Head	ft	0	10
4 (30.000)	30	60	Pressure Head	ft	0	10
5 (50.000)	60	90	Pressure Head	ft	0	10



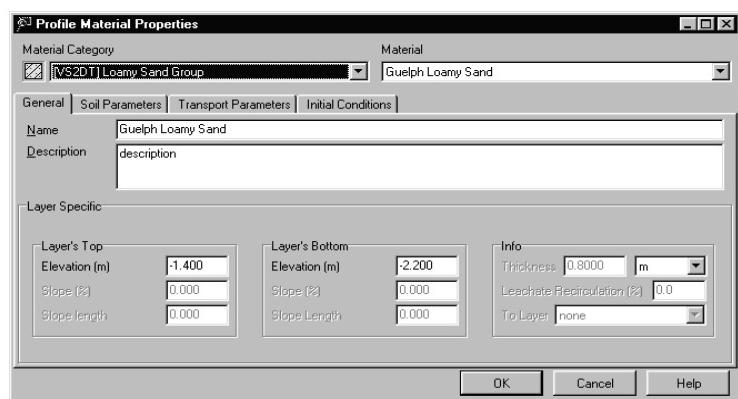
Now you may select the **Boundary type** and related **units** and edit the values of boundary conditions for each time period separately. The way to do it is quiet similar to that used for the **Case Settings** parameter group. The only difference is that units for specific type of boundary condition has to be uniform through the whole table (e.g. pressure heads for all periods has to be measured only in m or feet or cm).

The interface allows you to delete the wrongly set line or insert the line before the current. These operations can be done with **Insert** and **Delete** buttons.

Substituting Layer Materials and Chemicals

To substitute a layer or chemical and edit its parameters, you have to open **Profile Material Properties** dialog box.

☞☞ the name **Guelph Loamy Sand** from the example VS2DT profile tree structure. The following dialog box will appear:



To edit a specific group of parameters, click the appropriate tab.

In case you want to substitute the layer material, you have to select the appropriate **Material Category** and **Material**.

☞ the drop-down arrow of the **Material Category** text box. The list of available categories will appear:



Select a desired category and click it. Now you can open the list of materials available within the newly selected category and choose one of them.

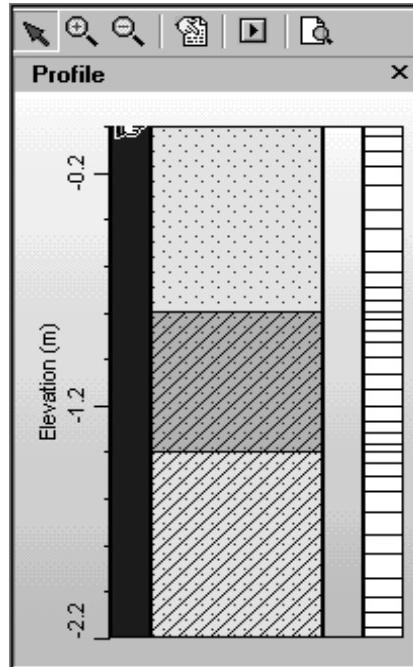
After you have selected the material, you may edit its parameters to make them completely matching your case.

These are the main common tools used to customize and edit profile objects in the Project Tree View. There are some other tools associated with specific models. These tools are described in the corresponding sections of the manual.

Working in the Profile View

The Profile View is a part of the interface where the profile is presented graphically and where its geometry and parameters may be edited.

A profile view of the default VS2DT profile is used to demonstrate common tools used in the Profile View window:



If you click anywhere within the profile picture, a menu will appear:



Through this menu you may perform operations with layer, view and edit profile properties, save this profile as a template for your future projects and print the profile picture.

You may use a number of graphical tools to modify profile structure to correspond with a desired or observed soil profile.

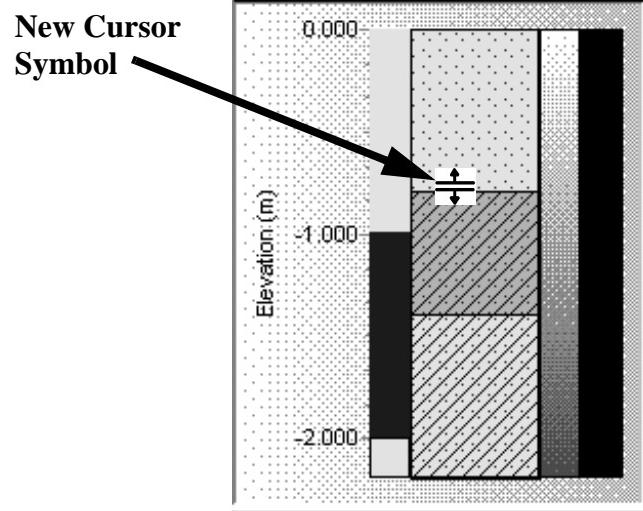
Because of the differences in nature of the simulated profiles, the sets of tools for profile editing in HELP and VS2DT differ and will be described in corresponding chapters. In this chapter the commonly used tools are explained. In both models the layering structure may be revised by either merging, splitting layers and/or resizing a layer. Merging layers essentially deletes one layer and expands the other layer into the space occupied by the deleted layer. Splitting layers divides a single layer into two layers. The properties for either layers may be changed to reflect the new desired profile.

Merging Layers

Layers can be merged by erasing their boundary. When you erase a layer boundary the layer that shares the boundary takes over the span of the area formerly occupied by the erased layer.

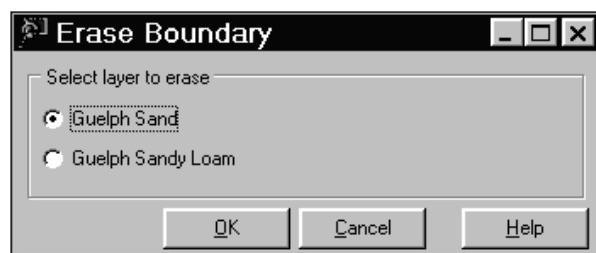
To erase layer boundary:

- 1) Move the mouse arrow between two layers. The cursor symbol will change:



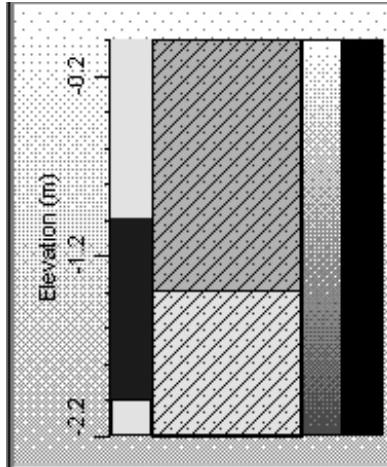
- 2) <right click>
- 3) Click **Merge Layers** from the appeared menu.

The **Erase Boundary** dialog box will appear:



- 4) Select the option button for the layer you wish to erase.
- 5) Click [**OK**].

6) The remaining layer will span the area once occupied by the deleted layer.



Restoring a Layer

You may restore the erased layer if you have not saved changes yet.

To restore a layer:

<right click> the layer's name in the Project Tree View.

2) Click **Restore**



Splitting a Layer

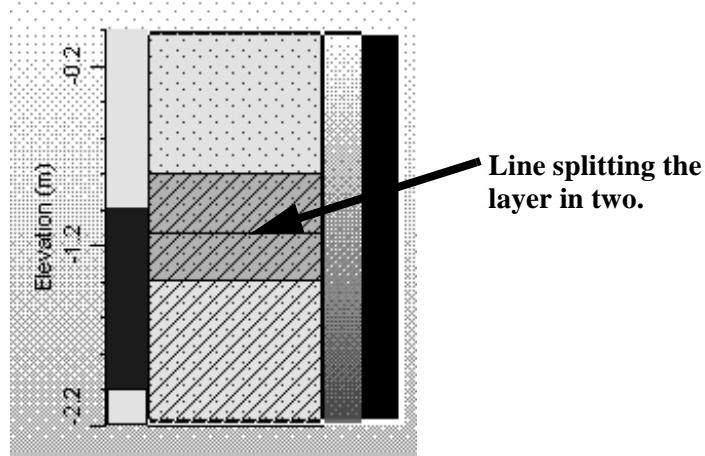
You can split a layer up into multiple sections and substitute material for each section or assign for each section different values for each parameter.

To split a layer:

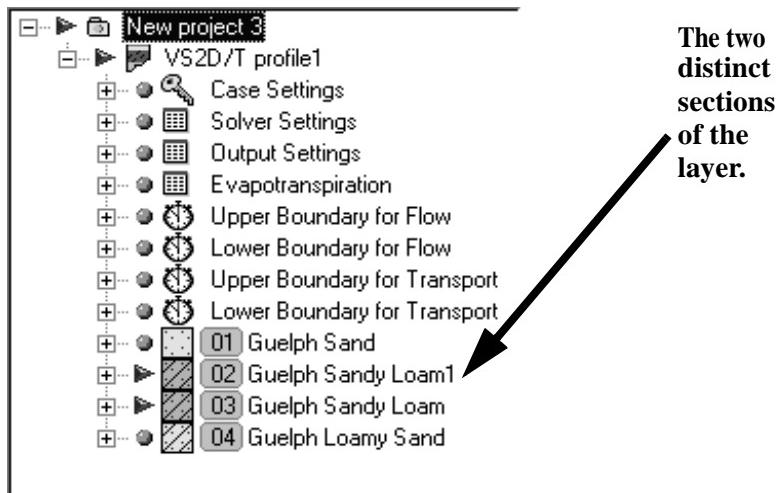
1) <right click> on the layer in the Profile View.

2)  **Layer/Split**

A line will appear through the layer at cursor position and a new layer will appear in the project tree.



Now each part of the layer can be edited separately with each section having its own unique properties. You may also substitute a material in the new layer.

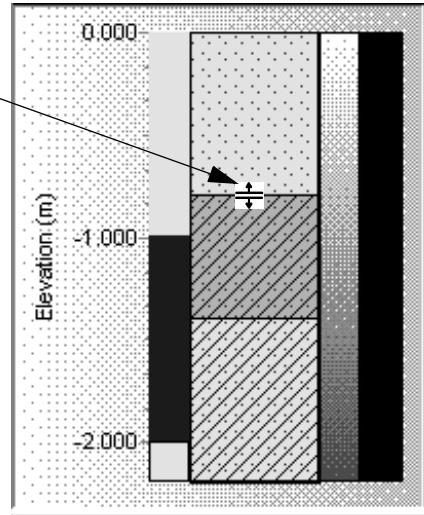


Resizing the Layer

To resize a layer:

- 1) Move the mouse arrow between two layers. The cursor symbol will change.

New Cursor Symbol



- 2) Click and drag the boundary to its new location.
- 3) Either accept the new elevation or type the correct elevation in the **Confirm Value** dialog box.
- 4) Click [OK].

Working with Multiple Profiles

To perform multi-variant simulations, UnSat Suite allows you to work with multiple profiles within a project. To get a new profile in a project, you may either open a default profile or copy one of already existing.



To open a default profile, click the Create New Profile icon in the Operational toolbar.

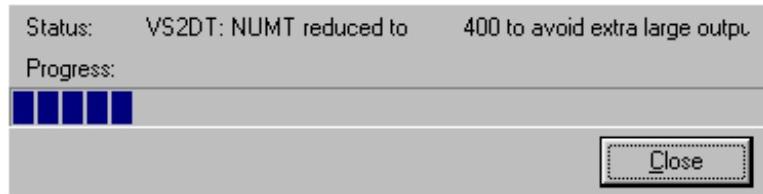
To copy an existing profile, <right click> the profile name. The following menu will appear:



Click **Copy**. A copy of the profile will appear in the Project Tree View and the picture of the new profile will replace the picture of the original profile in the Profile View. Now you may modify the newly added profile.

Running the Model

-  To run the model for a single profile click the profile icon above the Profile View.
 -  To run the model for multiple profiles or for one profile if it is single in the project, click the same icon in the Operational toolbar above the Project Tree View or  **Run** in the main menu and then click the model name.
- A progress bar will appear to indicate the computation progress:

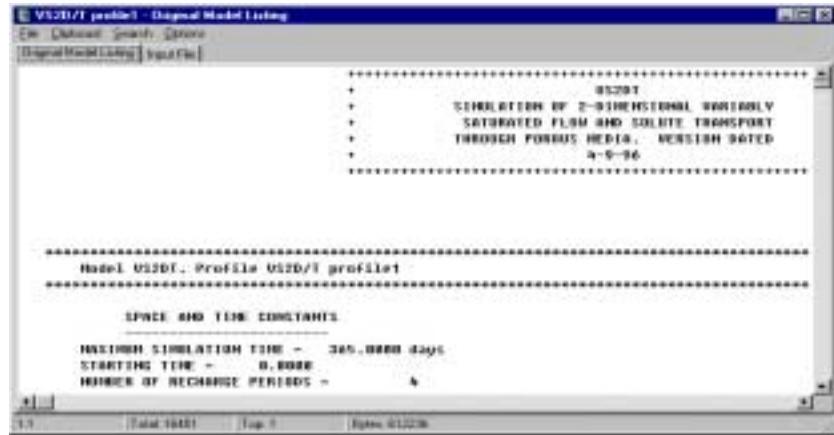


Viewing the Output

To view the original model output:

-  **Output** in the main menu, and then
-  **View Original Listing**

The following window will appear in case you are using the VS2DT model:

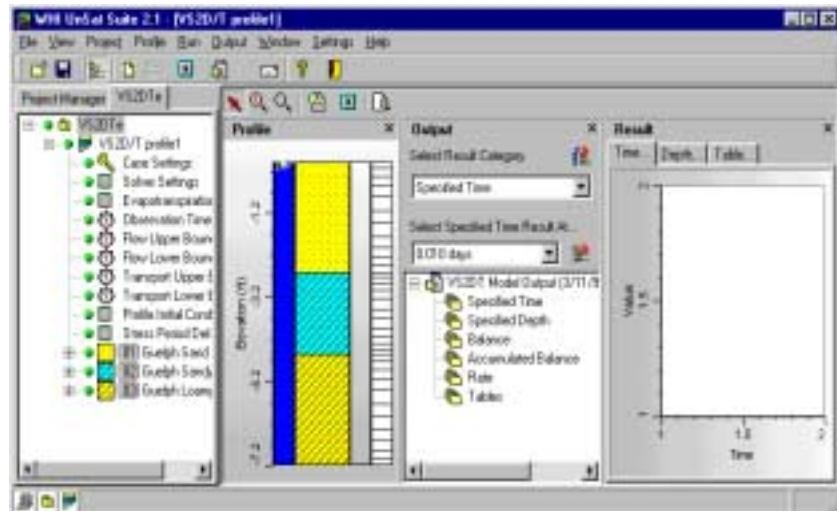


Here you can scroll and view the original listing, find specific expressions, mark and print parts of it or make a whole printout of the model results.

If you want to see the model's input file, click the **Input File** tab.

Viewing the Output Graphs

After the model has successfully run, the Output View and Result View windows will open and the UnSat Suite window will look the following way:



To enlarge the viewing area of the Result Window you may click the icon to close the Project Tree View or click the 'X' in the Profile View to close it.

To select the output category, click the arrow in the **Select Result Category** drop-down list box. The following list will appear in case you are using the VS2DT model:



Click the category you wish to view.

The first possible result group will appear in the list box below. To view all available result groups, click the arrow in the lower drop-down list box (**Select 'Name of Category' Result at...**).

The list of all arguments will appear drop-down in the list box. The picture below shows the case when **Specified Time** was chosen in the **Select Result Category box**:

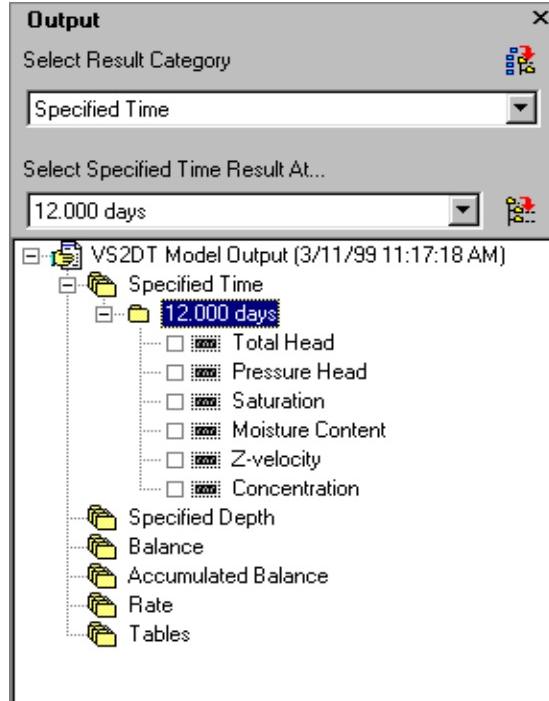


Use the slider to reach the time of your interest and click it. The selected time will appear in the drop-down box:

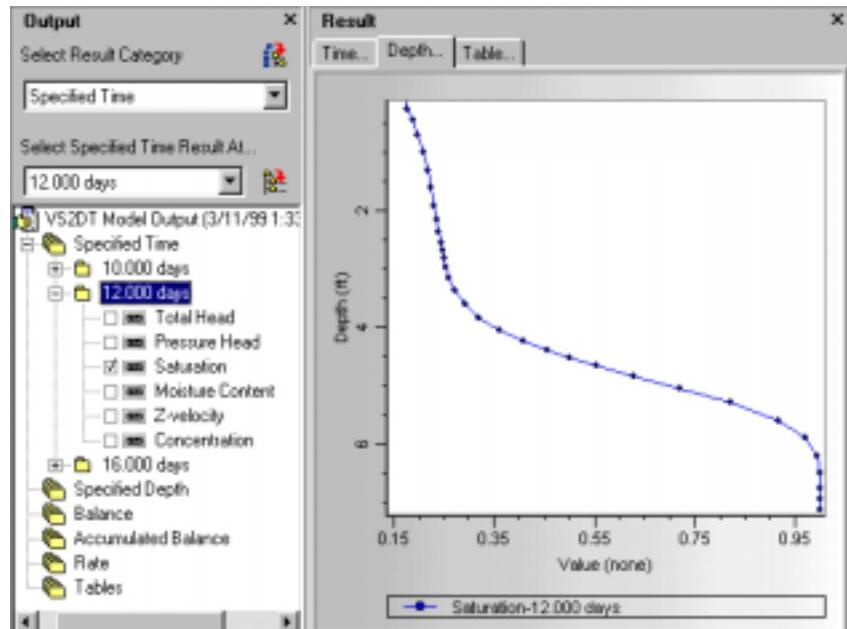


 To view all results available for this specific time, click the icon to the right of the **Select Specified Time Result at...** box.

The list of available results will open in the Result Tree:

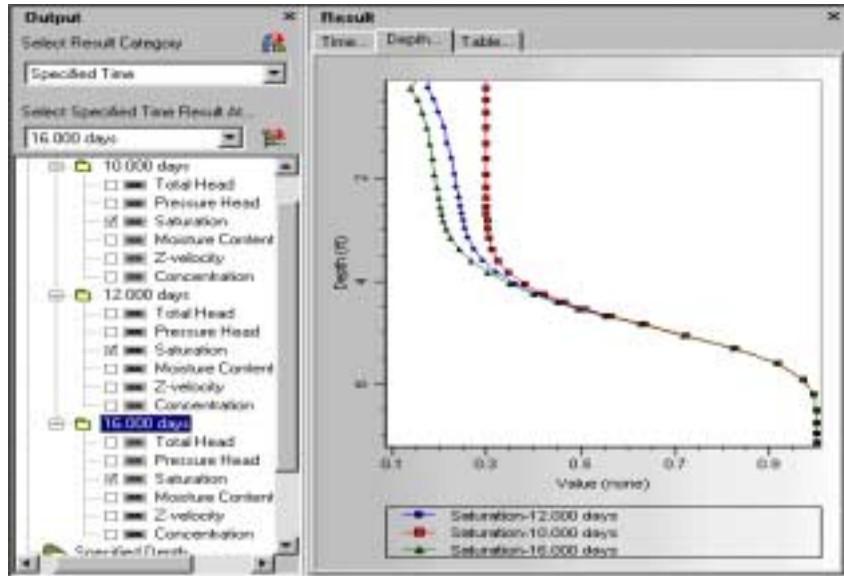


Click the check box beside the type of variable which you wish to view. The graph of the variable will appear in the Result View window:

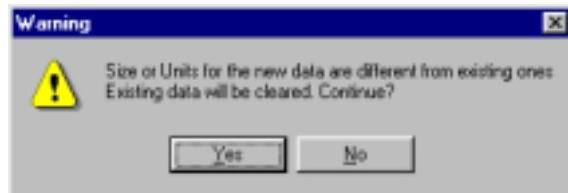


To add the graph for another time to the same window, select a new time from the **Select Specified Time Result at...** box and check the same variables (you will get a warning if you choose different variables). The

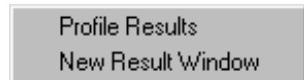
Results View will show profile distribution of the variable for different times:



If you wish to view output for another variable, click the corresponding check box. The warning will be posted if the new and previous variables are measured in different units:



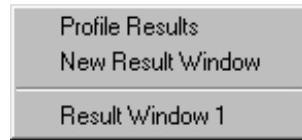
If you want to view both variables, you may place results for the second variable into the new Result Window. To do this, <right click> the name of the second variable. The following menu will appear:



Choose **New Result Window**. Results for the second variable will appear in the **Result Window 1**. You may add graphs for the other times to the **Result Window 1** using the same method.

If you want to see outputs for more than two variables, you may open additional Result Windows and place results there. To do this, <right

click> the name of the additional variable. The following menu will appear:



Choose **New Result Window**. Results for the second variable will appear in the **Result Window 2**. You may repeat these steps until you see everything you are interested in.

You may change units for your output variable without rerunning the model. From **Main Menu** select **Project/Properties** and than **Units** tab. In Units dialog box, select the proper Unit template and

☞ **OK.**

Next time when you open your output graph, it will appear with the new units.

Viewing Tables

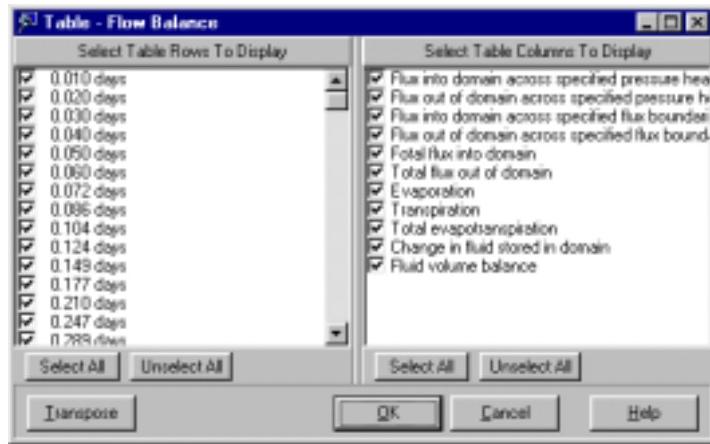
UnSat Suite allows you to view and edit output balance tables.

To access the desired table, click the arrow in the **Select Result Category** drop-down list box and click **Tables**. If you click the arrow in the lower drop-down box, you will get types of tables to select (the VS2DT model is used as an example):



Select the table from the list and click the icon to the right of the **Select Tables Result at...** box. The table will appear in the Output Tree. Click

the check box beside the table in the Output Tree to view all results available for this specific table. The following dialog box will appear:



Here you may select desired output times and variables to customize your table. You may use the following tools for editing a table:

- ☞ **Unselect All** to unselect all times or variables and then click desired if you want to show only a small number of rows or columns in the table.
- ☞ **Select All** if you wish to specify all list after you unselected some times or variables.
- ☞ **Transpose** if you want to switch columns and rows.
- ☞ **OK** after you have set a table.

The table will appear in the Result View.

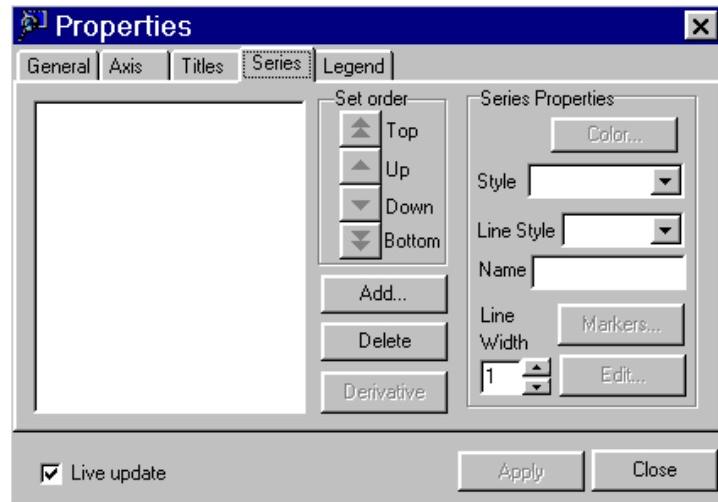
Editing Graphs

In the Results View you can plot the results of the simulation versus either the time of simulation or the depth of the profile. Once you have created a graph, you can edit it and save it as a bitmap in a separate file to include in reports and layouts.

UnSat Suite provides you with the flexibility to edit the layout of your graph directly in the Results View window. To edit your graph use the following method:

<right click> on the graph in the main window.

Properties



Above is the **Properties** window. This window has various features that allow you to customize your Unsat Suite Graphs. Below is a list of the tabs contained within the **Properties** window and the functions they perform.

Series Tab

Add or delete variables from the graph, position the variables on the graph, and change the style of the plot.

General Tab

Change the overall look of your chart by making it three dimensional, setting zoom preferences, initiating animation, adding a gradient to the plot, and enabling clip markers. You can also save the graph as a bitmap.

Axis Tab

Change the characteristics of the axes. You can show all, none, or some of the 'x' and 'y' axes. You can change the scales to logarithmic or inverted scales. You can also change the minimum and maximum values of the domain and range and add or edit titles, labels, and tick marks on the axis.

Titles Tab

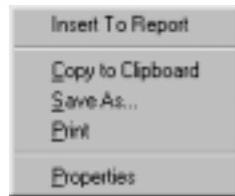
Change the graph title, and its characteristics. Add a footer to the graph.

Legend Tab

Change the position, framing, and style of the legend.

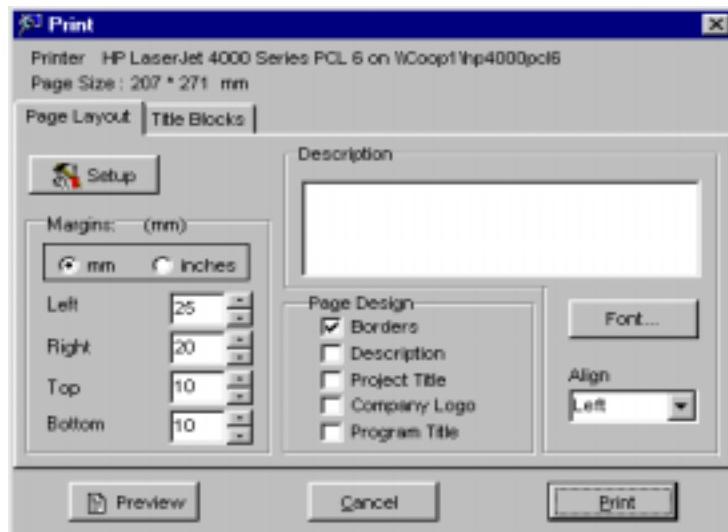
Printing an Individual Graph

To edit and print an individual graph, <right click> anywhere in the Result View area. The following menu will appear:



☞ Print.

The **Print** dialog box provides you with a number of options for printing UnSat Suite graphs.



[Preview]

Opens the **Print Preview** window. You can use the mouse to drag the graph to a new position or change the margins.

[Printer Setup] Opens **Print Setup** window.

[Print] Prints the selection.

[Exit] Returns to **Print** window.

[Setup] Opens the **Print Setup** window. You can adjust the paper size and source, choose a printer, and click either landscape or portrait style. In this window:

[Properties] Opens the printer's **Properties** window.

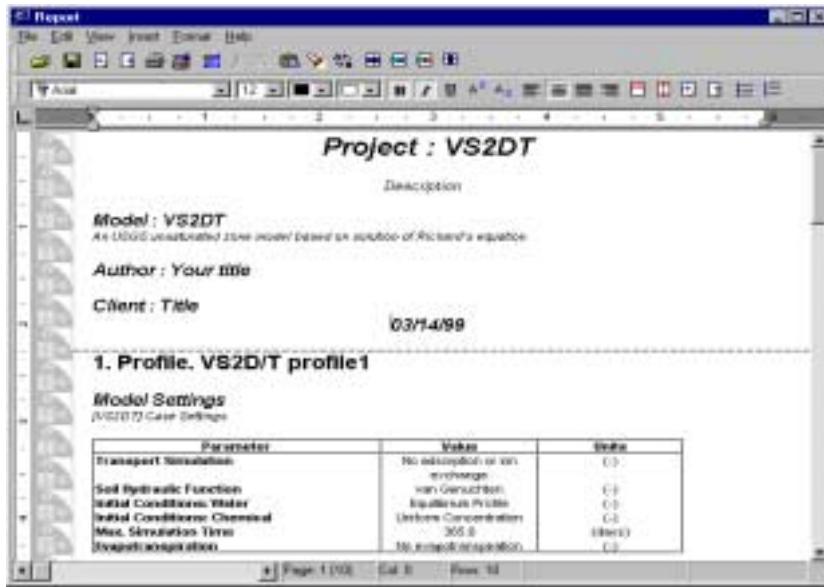
	[OK]	Saves changes and returns to Print window.
	[Cancel]	Discards changes and returns to Print window.
Include in Print		Click to include the project title and/or the company logo on the graph.
[Design]		Opens the Title and Logo Dialog window. Choose to design the project title or the company logo.
	Font	Choose font, style and size.
	Text	Type text that will appear on logo or title.
	Bitmap	Enter the bitmap you wish to open.
	[Delete]	Delete the selected graphic.
	[Browse]	Choose a graphic from file.
	[Save]	Saves design and returns to Print window.
	[Cancel]	Discards changes and returns to Print window.
	Margin	Change the page margins.
Margins Units		Choose to set the margins in mm or in.
Description		Type in an appropriate comment.
[Print]		Print the selected output.
[Cancel]		Close the Print window without saving the settings.
	[Help]	Shortcut to On-Line Help.

Preparing a Report

To present results of your VS2DT simulation to your clients you may use the UnSat Suite Report Generator.



To create a report and add the project input data to it, click the icon from the Operational Icons toolbar. The report will appear in a separate window:

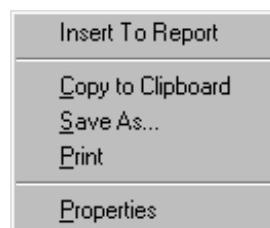


In this window you may edit the report, input your own text and add any types of graphic or table output produced by UnSat Suite.

Note: the graphs and tables will be placed at insertion point.

To add a graph or a table to the report:

- [1] In the **Report** window place the cursor to position where you want your graph or table to appear in the report
- [2] Create a graph or table using one of the methods described above and
- [3] <right click> in the Result View. The following menu will appear:



- [4] **Insert To Report.** The graph or table will appear in the report.

The graph may appear smaller than the original. To get the graph of desired size, click it in the **Report** window and stretch it until it will reach the proper size.

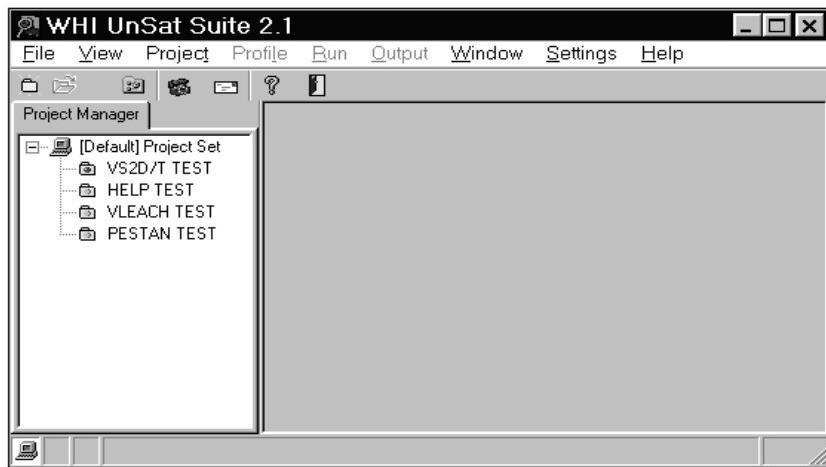
4

Working with Project Sets

Definition of the Project Set

When UnSat Suite is opened, it reveals a Default Project Set in the Project Tree which is a collection of WHI UnSat Suite projects. In addition to the default project set, you can archive completed project sets or open and use project sets stored on your machine or other machines within the local area network.

An example of Default Project Set is shown below:



The Default Project Set is located on the user's computer and shows a computer icon next to the title. Next to each project is a project folder icon. This icon has a green light (light grey circle in the black and white caption above) in the center to indicate the file is not open and is accessible.

Using the Project Set Manager

In the Project Set Manager, all project sets are clearly displayed with their paths and alias name. All archived project sets will also appear in this window.

To open the Project Set Manager:

From the **File** menu **Project Set Manager**. The following dialogue box will appear:



The main menu contains the following components:

File Create a new alias, delete selected aliases, close the dialogue box, or select all of the aliases.

Help Open UnSat Suite on-line help.

The following icons form the toolbar of the **Project Set Manager**.



Create New Alias

Map onto a remote project set through a network connection.



Select All

Select all aliases. Selected aliases show a checkmark in the box adjacent to their name.



Delete

Delete all selected aliases.



Help

Open on-line help.



Close Dialogue Box

Close the dialogue box.

Deleting Alias

When you delete in Project Set Manager you do not delete the project set, you only delete the mapped connection to the project set. The deleted path to the project can be mapped again if needed.

To delete a project path:

- 1) on the check box for the project(s) you wish to delete.
- 2) 

To delete all project paths:

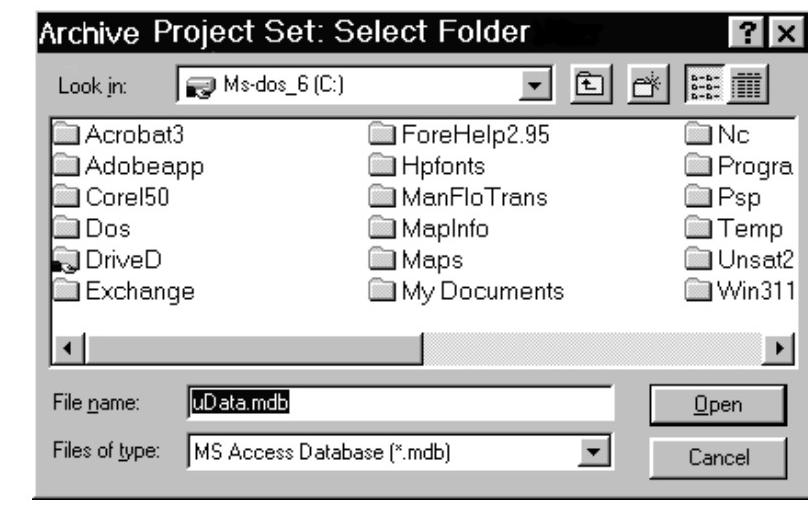
- 1) 
- 2) 

Archiving and Copying Project Sets

Once you have completed a set of projects in your default project set, you can archive the completed set. The archived set can still be opened and edited at any time, but the files will not appear in the default project set. Therefore, authors can organize their projects without having a cluttered work space.

To archive or copy a project set:

- 1) Close all projects and return to the Project Manager.
- 2) From the **File** menu  **Copy Project Set**.
- 3) The **Copy Project Set: Select Folder** dialogue box will open.



- 4) Select the folder where you would like to save your completed project set.
- 5) In the **File name** box, type the name of the project set you are about to store. Make sure it is a unique name or it will automatically override an existing file with the same name.  **[Open]**.
- 6) You will be prompted to create the new file.  **[Yes]**.
- 7) The **Confirm Database Alias** dialogue box will open. In the **Alias** box, type the name of the project set. This is the name you will use to open the project set in UnSat Suite.
- 8)  **[OK]**. The stored project set will become the current project set in the Project Tree.

Returning to the Default Project Set

To return to the Default Project Set:

- 1) From the **File** menu  **Reopen Project Set**.
 - 2)  **Default**.
- OR
- 1)  
 - 2)  **Default**.

Deleting Projects from the Default Project Set

Once you have returned to the Default Project Set you can delete the projects that you have just archived.

To delete a project:

- 1) <right click> on the project in the Project Tree.
- 2)  **Delete**.

To delete all the projects:

- 1) <right click> **[Default] Project Set**.
- 2)  **Delete All Projects**.

After the projects are deleted you can create new projects in your empty project set.

Opening an Archive Project Set

The Archive Project Set can be opened and edited at anytime.

To open an archived project set:

- 1) From the **File** menu  **Reopen Project Set**.
- 2)  the project set name.

Opening a Local Project Set

Project sets can be opened in one of two ways:

To open a project set:

- 1) From the **File** menu  **Reopen Project Set**.
- 2)  the project set name.

Or

To open a project set:

- 1) 
- 2)  the project set name.

Working over a Local Area Network

In addition to opening stored project sets, you can map onto a project set through a network connection. This allows you to conveniently view and edit a colleague's work.

To access a project set through the local area network, use one of the following methods:

- 1) 
- 2)  **Open**

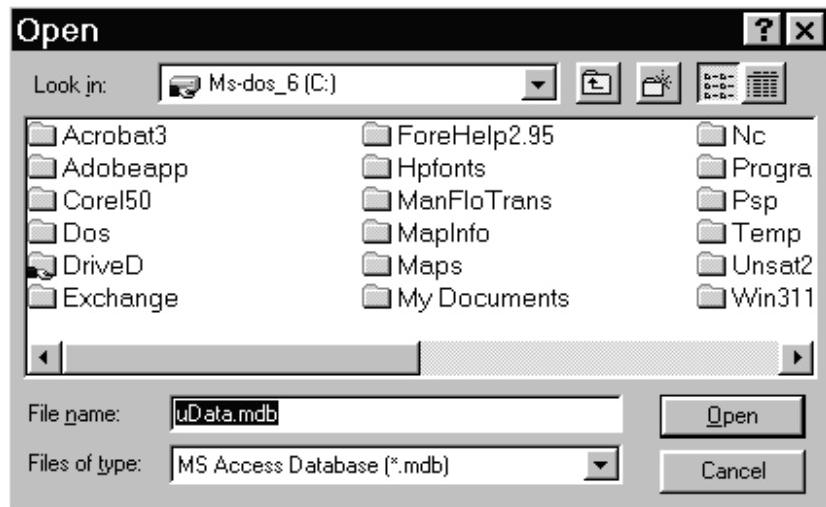
Or

- 1) From the **Settings** menu  **Project Set Manager**. The **Remote Data Alias Manager** dialogue box will appear.
- 2) From the **File** menu  **New Alias**.

Or

From the **File** menu  **Open Project Set**.

The **Open** dialogue box will appear:

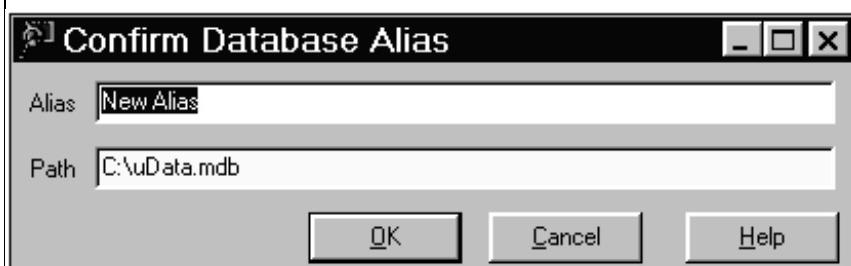


From here you may access the remote project set:

- 1) Navigate to **My Computer\Desktop**.
- 2) Select **Network Neighborhood**.
- 3) Select the computer on which the desired project set is located.

To map onto a network project set:

- 1) Locate the project set you wish to map.
- 2)  **[Open]**. The **Confirm Database Alias** dialogue box will open:



- 3) Type the name of the project set in the **Alias** box.
- 4)  **[OK]**.

The project set can now be edited and viewed on your computer.

Copying a Project From One Project Set to Another

You can copy a project contained in one project set over to another project set even if that project is on another machine. You can copy a project from one project set to another with one of the following methods:

To copy a project from one project set to another:

- 1) From the **File** menu **Reopen Project Set**.
- 2) the project set which contains the original project. The project set will open.
- 3) Click on the project to copy in the Project Tree and drag the project to the storage trunk icon:



- 4) The copy holder icon will change its view. This indicates



that the copy holder is holding the project.

- 5) From the **File** menu **Reopen Project Set**.

- 6) the project set you want to copy the file to.



- 8) the project name that you wish to paste in the current project set.

- 9) Enter or confirm the name of the project and click **[OK]**,

Or

To copy a project from one project set to another:

- 1) From the **File** menu **Reopen Project Set**.

- 2) the project set which contains the original project.

- 3) <right click> on the project in the Project Tree.

- 4) **Copy...**

Note: If you choose Make Copy, the file will be copied to the current project set and you will be prompted to give it a distinguishing name.

- 5) From the **File** menu **Reopen Project Set**.

- 6) the project set you want to copy the file to.



- 8) the project name that you wish to paste in the current project set.

- 9) Click **[OK]**. The project will appear in the new project set.

Note: You can copy more than one project at a time. Use Paste All in the copy holder to paste all the copied projects into the current project set.

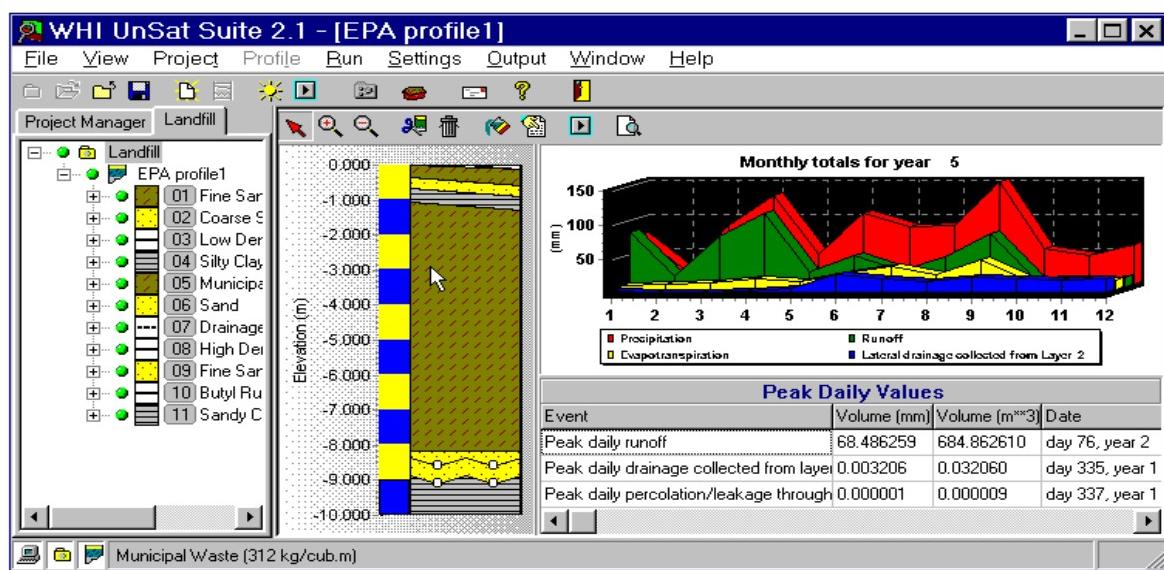
Repairing the Project Set

Because of data overflow or system failures the crashes of the WHI UnSat Suite projects may happen at times. When you open the WHI UnSat Suite project set again, the damaged project will be marked with the red bullet in the Project Tree View.

To repair the project set:

- 1) Select the project set name in the Project Tree View.
- 2) In Main menu open **Settings**, and
- 3)  **Repair**. The red bullet will turn green. You may continue working with your project after performing this operation.

Part 2: The HELP Model



Introduction

HELP is a versatile model for predicting landfill hydrologic processes and testing the effectiveness of landfill designs, therefore, enabling the prediction of landfill design failure resulting in groundwater contamination. HELP has become a requirement for obtaining landfill operation permits in the U.S. HELP is also effective in assessment of groundwater recharge rates.

The quasi-two-dimensional hydrologic model accepts the following input data:

- Weather (precipitation, solar radiation, temperature, evapotranspiration parameters)
- Soil (porosity, field capacity, wilting point, and hydraulic conductivity)
- Engineering design data (liners, leachate and runoff collection systems, surface slope)

The profile structure can be multi-layered, consisting of a combination of natural (soil) and artificial materials (waste, geomembranes) with an option to install horizontal drainage, and change the slope of profile parts (e.g. landfill cap, leachate collection and removal systems).

HELP uses numerical solution techniques that account for the effects of surface storage, snowmelt, runoff, infiltration, evapotranspiration, vegetative growth, soil moisture storage, lateral subsurface drainage, leachate recirculation, unsaturated vertical drainage, or leakage through soil, geomembrane, or composite liners.

Built-in Databases and tools:

- Weather Generator, a tool for synthetic generation for up to 100 years of daily values of precipitation, air temperature and solar radiation.
- Soil, waste and geomembrane database which contains parameters for 42 materials.

History

WHI has developed a Windows 95/98/NT interface for the version 3.08 of the HELP model which was released in May 1998 under the name Visual HELP version 1.101. This software was presented at the trade show at SWANA's WASTECON 1998 and ISWA World Congress 1998 in Charlotte, North Carolina. Currently, more than 300 copies of this product are being used by consulting companies, government regulator bodies and Universities in U.S.A., Canada, U.K., Germany, Australia, Sweden, Mexico, France, Slovenia, Slovakia and Hungary.

New Features of Visual HELP 2.1

Profile Viewing and Editing:

- A layer now can be split into two separate layers
- Numbers of the layers appear in the Project Tree, which simplifies interpretation of results

Weather Generating:

- New databases and GIS searching tools have been developed for the major regions of the world (more than 3000 weather stations)
- Data in NOAA format now can be imported automatically
- Diagnostics is provided to detect missing records in NOAA files

Output Presentation:

- Output units can be changed now without restarting the project
- Water Balance Tables have been added
- New tree-like interface structure allows selecting results for displaying more efficiently
- Report Generator (a new feature of Visual HELP) allows you to display, print and export to Microsoft Word project input parameters and settings and output graphs and tables.

New Features of Visual HELP 2.2

Visual HELP version 2.2 allows you to easily transfer data between the WHI UnSat Suite models, as well as export model data for use with other applications (e.g. export groundwater recharge assessed by Visual HELP to Visual MODFLOW).

Some users might wish to export Visual HELP simulation results and import them into post-processing spreadsheet programs (e.g. MS EXCEL). Version 2.2 is also capable to perform this operation.

Importing a Visual Help 1.1 model into Visual Help 2.2

To import a Visual HELP 1.1 project, click on File/Import Help 1.1 from the Main Menu. The Import Visual Help 1.1 Projects window will open, and shown in the following figure:



Click the Browse button to locate your Visual HELP 1.1 project, and use the checkboxes to select the appropriate options.

Known Limitations:

Some models may import without the Drainage Setting for Drainage Type Layers. Please manually verify that this is set using the Layer Properties Dialog of Visual HELP.

5

Designing the Landfill Profile

Profiles in Visual HELP

For the purpose of hydrological simulation, a profile represents a part of a landfill that is assumed to have the same cross-section throughout. The profile contains all of the layers of the landfill that it represents. It may also contain details of engineering components, such as subsurface drainage, leachate recirculation systems, geomembranes, geonets, and composite liners. The surface of the profile can be sloped, which is typical for the peripheral parts of the landfill, or flat, which is typical for the central part of the landfill. The slope of the landfill usually mimics the shape of the waste body. At the time of landfill closure, the waste layer at the periphery of the landfill is usually sloped and the entire landfill site is capped by several layers, which form the landfill cap. Although drainage pipes are not usually installed in the landfill cap, permeable sand layers in the cap can transport water to the bottom drain or sump that surrounds the landfill. This process can be simulated by UnSat Suite.

A typical landfill can be represented as a set of profiles. This is done by creating several profiles in one project. This is useful because it allows you to use one profile to simulate the middle of the landfill, and several other profiles to simulate the edges, where the cross-section is tapering.

Since the size of a landfill is much smaller than the distance at which weather changes, the same set of weather data is used for all of the profiles in a project.

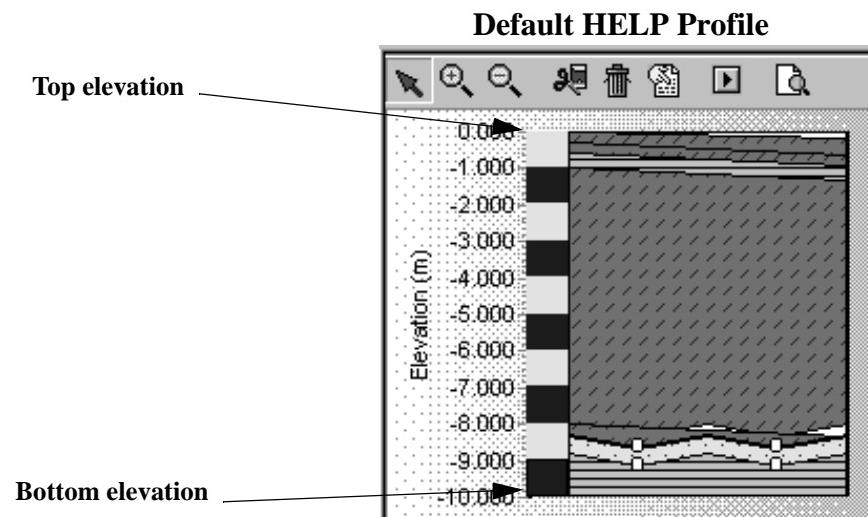
In the current version, the landfill water balance is calculated separately for each sub-area, which is represented by a single profile, but future releases of UnSat Suite will allow you to integrate the water balance across the entire landfill by calculating the water balance for the entire project.

The natural water balance components that UnSat Suite simulates are precipitation, interception of rain water by leaves, evaporation from leaves, surface runoff, evaporation from soil, plant transpiration, snow accumulation and melting, and percolation of water through the profile. Data, representing meteorological conditions, can be imported from a file or synthetically simulated with the Weather Generator.

Before running the simulation, the initial water content of different landfill layers should be specified. As with the original DOS HELP, UnSat Suite gives you the option to have the initial water content values specified by you or computed by the model (as nearly steady-state values). With the latter, which is the default, UnSat Suite will assign realistic values for the initial moisture storage of layers and simulate one year of landfill hydrology. The values of moisture storage obtained from this simulation will then be used as initial values, and the simulation will start again from the beginning of year 1.

The HELP model is typically used to simulate the failure of a waste contaminant design. Civil engineers use this model to try different landfill designs and find the optimal combination of the landfill performance and cost. The thickness of individual layers and the total profile depth are the terms of the optimization equations and can be edited by the user at any time.

Using the HELP model, UnSat Suite allows the user to fix the top elevation or the bottom elevation of the designing profile. The top elevation can be fixed when there is a constraint on the appearance of the landfill. The bottom elevation can be fixed when there is a constraint on the depth of the landfill (e.g. the bottom of the landfill should be above the highest groundwater level). When the user edits the thickness of a layer, the profile will change differently, depending on whether the top or bottom elevation was fixed in the Profile Properties dialog box. If the profile top was fixed and the user increase thickness of a layer, the profile will grow downwards. If the profile bottom was fixed and the user increase the thickness of a layer, the profile will grow upwards.



Layers in the HELP model are organized by the hydraulic function that they perform. Four types of layers were available in the original DOS

HELP: vertical percolation layers, lateral drainage layers, barrier soil liners, and geomembrane liners. Topsoil and waste layers are generally **vertical percolation layers**. Sand layers above liners are typically **lateral drainage layers**. Compacted clay layers are typically **barrier soil liners**. Geomembranes form the group of **geomembrane liners**. Composite liners can be modeled as two layers: a barrier soil layer, and a geomembrane. In the original DOS HELP, geotextiles were not considered as layers unless they performed a unique hydraulic function. In UnSat Suite, **geotextiles and geonets**, which are a class of landfill material that is growing extensively, are recognized as a separate type of layer.

The graphical representation of each type of layer in a profile is as follows:

-  Vertical Percolation Layer
-  Lateral Drainage Layer
-  Barrier Soil Liner
-  Geomembrane Liner
-  Geotextiles and Geonets

The symbol for each layer is the same in the Project Tree as it is in the Profile View, except for the Geomembrane Liner and the Geotextiles and Geonets. These two layers appear as follows:

-  Geomembrane Liner
-  Geotextiles and Geonets

Subprofiles

The HELP model calculates water flow by subprofiles. A subprofile is a set of layers that has a barrier soil or a geomembrane liner at the bottom. At times, a real world profile which you wish to simulate may have a highly permeable material of **Vertical Percolation** or **Lateral Drainage** material category at the bottom. In this case, we recommend you to split the bottom layer into two and do a formal change with the lower part (5-10 cm thick): specify it as a **Barrier Soil Liner** remaining parameters the same as for the upper part. From hydraulic point of view, the new profile is the same as the original one. However, the formal rule of HELP code will be satisfied and result of simulation will be more accurate.

Layering Rules

Although UnSat Suite allows a wide range of layer combinations, there are some basic rules for arranging layers in profiles. These rules are summarized in the following table:

Layer Type	Forbidden combinations			Maximum number of layers in profile	Can it be a top layer?
	Above it	Below it	Between		
I. Vertical Percolation Layer	II and V cannot be above I				Yes
II. Lateral Drainage Layer		I cannot be below II			Yes
III. Barrier Soil Layer	III cannot be above another III	III cannot be below another III	III cannot be between a IV and another IV	5	No
IV. Geomembrane Liner	IV cannot be above another IV	IV cannot be below another IV	IV cannot be between a III and another III	5	No
V. Geotextiles and Geonets		I cannot be below V			Yes

Visual HELP will warn you if you try to delete or insert a layer incorrectly.

Note: If two Lateral Drainage or Geomembrane layers are adjacent, the drainage parameters for the lower layer only will be used for calculation.

Profile Properties

Profiles contain more information than simply the number and types of layers contained in the profile.

To change the profile properties, <right click> the profile in the Profile View, and click **Profile Properties**

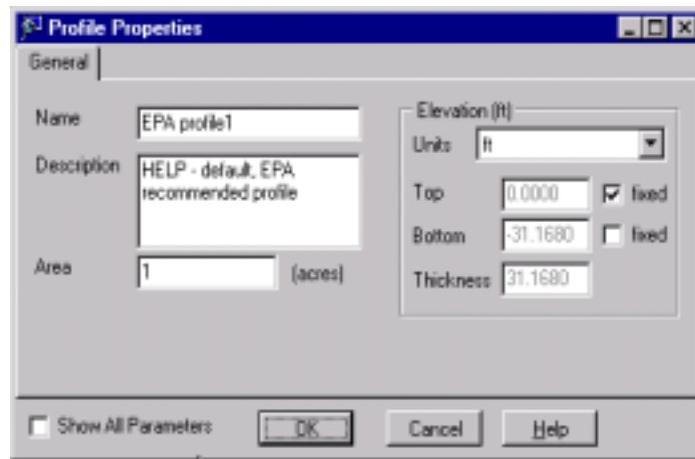
Or

<right click> the profile name in the Project Tree View,

Or

<right click> the profile name in the Project Tree View, and click **Properties**.

The **Profile Properties** dialogue box will appear:



Name	Type the name of the profile in this box.
Description	Type a description of the profile here.
Elevation	If the Top fixed check-box is selected, you can set the maximum height of the profile. Any editing of the profile's thickness will result in the bottom elevation to change.
Area	If the Bottom fixed check-box is selected, you can set the maximum depth of the profile. Any editing of the profile's thickness will cause the top elevation to change.
	Type the land area represented by the profile in this box.

Case Settings

The **Case Settings** parameter group of the profile contains parameters which affect the entire landfill. The factors found in **Case Settings** determine how the model calculates the amount of water runoff from the landfill and how much water is initially present in the landfill. Depending on the selection made, runoff Curve Number can be calculated by the model, which requires no additional input by the user, or the runoff Curve Number can be specified or calculated by the user. In both of the latter cases, the user must input the runoff area and the runoff curve number.

When determining a calculation method for the initial moisture settings, the user can specify not only whether or not the model will calculate the value, but whether or not the landfill will have a uniform moisture throughout. Each individual layer can have a different amount of initial moisture storage. This is specified in the Profile Properties dialogue box. In Case Settings, the user only indicates how the model is to calculate the storage amount. Indicating user specified allows the user to select uniform or non-uniform concentrations of initial water storage later on.

Editing Case Settings

Editing **Case Settings** can cause changes in other parameters depending on the selection made in the Case Settings.

To edit case settings:

- 1) <right click> on **Case Settings**.
 - 2) Click **Edit**. The **Edit Parameters** dialogue box will open,
- Or**
- ☞ the **Case Settings**.
- 3) From the **Value** list click your selection.
 - 4) Click **[OK]**.

Runoff Method

As with the original DOS HELP, the rainfall-runoff processes in UnSat Suite are modeled using the USDA Soil Conservation Service curve-number method, which is widely accepted and allows you to adjust the runoff calculation to a variety of soil types and land management practices. The curve-number method was developed using rainfall-runoff data for intensive storms on small watersheds.

The curve number (CN) is defined with respect to the runoff retention parameter (S), which is a measure of the maximum retention of rainwater after runoff starts (in inches).

$$CN = 1000 / (S + 10)$$

The maximum value of CN, which is 100, occurs when there is no infiltration. The smaller CN is, the more rainwater will infiltrate into soil. The minimum realistic value for CN can be assumed to be approximately equal to 50. UnSat Suite uses different procedures to adjust the value of CN to surface slope, soil texture, and vegetation class.

The model will calculate the CN for all runoff selections except for **User Specified** and **User Modified**. Selecting one of these allows you to enter your own CN number.

There are three options available: **Model Calculated** (which is the default), **User Specified**, and **User Modified**. Clicking anyone of these will determine what parameters will need to be specified further. The following chart summarizes the parameters that become active after clicking an option:

Selection	Location of New Parameter	New Parameter
Model Calculated	Surface Water Settings	Runoff Area = Percentage of surface area with runoff. Vegetation Class = Class of vegetation cover.
User Specified	Surface Water Settings	Runoff Area = Percentage of surface area with runoff. Runoff Curve Number = USDA SCS CN method.
User Modified	Surface Water Settings	Runoff Area = Percentage of surface area with runoff. Runoff Curve Number = USDA SCS CN method.

Initial Moisture Settings

As with the original DOS HELP model, UnSat Suite allows two possibilities for setting the initial moisture storage. You can select either **Model Calculated** (which is the default), or **User Specified**. If you select **Model Calculated**, HELP will assign realistic values for initial moisture storage and then simulate one year of landfill activity. The values of moisture storage obtained from this simulation will be used as the initial values. The following chart shows the new parameters which become introduced by either selection:

Selection	Location of New Parameter	New Parameter
Model Calculated	Surface Water Settings	none
User Specified	Surface Water Settings Lateral Drainage Layer, Vertical Percolation Layers, and Geotextiles and Geonets	Initial Surface Water = Initial amount of water at the soil surface (snow0). Initial Moisture Content = Initial moisture content for the layer.

Editing the Surface Water Settings

Surface Water Settings can be edited using the following method:

To edit the Surface Water Settings:

- 1) <right click> on **Surface Water Settings**.
 - 2) Click **Edit**. The **Edit Parameters** dialogue box will open,
- Or**
- 3) In the **Value** box type the new value.
 - 4) Click the units from the **Units** list.
 - 5) Click **[OK]**.

Runoff Area

You are always able to change the percentage of the profile's surface area that has runoff.

Vegetation Class

Choose the surface vegetation from this list. Vegetation has a significant effect on the runoff pattern. You can click one of the following:

- Bare Soil
- Poor Stand of Grass
- Fair Stand of Grass
- Good Stand of Grass
- Excellent Stand of Grass

Initial Surface Water

Initial amount of water at the soil surface in form of snow or ice.

Editing Landfill Layers and Modifying the Profile

As it was described at the beginning of the chapter, all layers in Visual HELP are identified by the hydraulic function that they perform. The following design layer categories are available:



Vertical Percolation Layer

Usually this is a topsoil, suitable for vegetative growth, or a waste layer. The primary purpose of a vertical percolation layer is to provide moisture storage.



Lateral Drainage Layer

Usually this is a material with moderate to high permeability (e.g. sand or gravel) that is underlaid by a liner with a lateral drainage collection and removal system. The primary purpose of a lateral drainage layer is to transport water towards the drainage pipe.



Usually this is a soil with low permeability (e.g. loam or clay), often compacted, which is designed to limit percolation and leakage.



This is a synthetic flexible membrane designed to restrict vertical drainage, and limit leakage.



This is a new type of layer that was not available in the original DOS HELP.

It is a synthetic material designed to drain water laterally. Geotextiles and geonets are produced industrially and have specific thicknesses, which are determined by the limitations of industriously technologies. Whereas the thickness of lateral drainage layers can be specified by the user in UnSat Suite. Geotextiles and geonets are not resizable in UnSat Suite.

Layer Properties

All of the layers have the same parameters, except for the **Geomembrane Liner**. The common parameters are described below:

Total Porosity	Average ratio of empty space to material.
Field Capacity	Moisture storage after a prolonged period of gravity drainage, corresponding to a suction of 1/3 bar.
Wilting Point	The lowest moisture storage that can be achieved by plant transpiration or air drying, corresponding to a suction of 15 bars.
Saturated Hydraulic Conductivity	Permeability of saturated material under a unit pressure gradient. A unit pressure gradient refers to a unit change in head (measured in units of length) per unit change in distance downstream (in the same units of length), in saturated 1-D flow. Note that a Geomembrane is considered to have Isotropic Saturated Hydraulic Conductivity,

	therefore only 1 value is required for both lateral and vertical Hydraulic Conductivity.
Subsurface Inflow	Inflow from an external source into the layer. If subsurface inflow is assigned to the bottom layer, no leakage through the bottom layer will be simulated.
Initial Moisture Content	Initial moisture content for the layer. This is only present when the initial moisture setting is specified.
Geomembrane Liners have saturated hydraulic conductivity , as well as a number of other special parameters, which are described below:	
Pinhole Density	The number of defects per unit area resulting from manufacturing flaws. It is assumed that the diameter of the hole is equal to, or smaller than, the geomembrane thickness. Holes are estimated to be one millimeter in diameter.
Installation Defects	The number of defects per unit area as a result of the installation. It is assumed that the diameter of each hole is larger than the geomembrane thickness. Holes are estimated to be one square centimeter in area.
Placement Quality	Quality of contact between the geomembrane liner and the undersoil. This parameter can take on the following values:
1. Perfect:	Perfect contact between geomembrane and adjacent soil.
2. Excellent:	Exceptional contact between geomembrane and adjacent soil, which limits drainage rate (achieved only in laboratory or field experiments).
3. Good:	Good field installation with smooth soil surface and geomembrane wrinkle control.
4. Poor:	Poor field installation with a less well-prepared soil surface and/or geomembrane wrinkling, which provides poor contact between the geomembrane and the adjacent soil, resulting in a large gap for spreading and greater leakage.
5. Worst Case:	Contact between geomembrane and adjacent soil does not limit drainage rate; leakage rate is controlled only by the hole.

6. Geotextile separates geomembrane and drainage limiting soil:

Leakage rate is controlled by the in-plane transmissivity of the geotextile.

Geotextile Transmissivity The product of the saturated hydraulic conductivity and the thickness of the geotextile. This parameter should be specified only when the placement quality is six.

Editing Layer Properties

You can edit the properties of a layer that already exists in the profile.

Open the **Edit Properties** dialogue box, by using one of the following methods:

☞☞ the layer in the Profile View,

Or

☞☞ the layer in the Project Tree View,

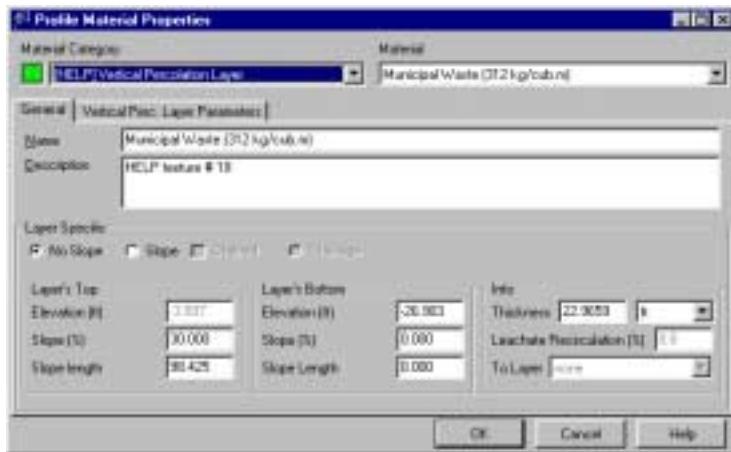
Or

<right click> the layer in the Project Tree View, and click **Properties**,

Or

<right click> the layer in the Profile View, and click **Layer Properties**.

The **Edit Properties** dialogue box is shown below:



General Tab

Under the General tab, you can edit properties of the layer. These include **Design Layer Category** and **Material**, **Slope** and **Drainage**, and several other layer parameters. **Drainage** and **Sloped [Drained]** are available only for Lateral Drainage Layers, and Geotextiles and Geonets.

To change the **Design Layer Category** or **Material**, click the list and click on your selection. Click [OK]. If you change the **Design Layer Category**, you will also need to specify a new **Material**.

Note: Changes to Design Layer Category are limited to layers that are allowed at the insertion point.

To change the slope and drainage properties of the layer, click one of the available buttons: **No Slope**, **Slope (Drained)**, or **Drainage**. Click [OK] to save your changes. The **Drainage** button applies in cases where drainage pipes exist. If there are no pipes, then click either the **No Slope** button or the **Slope** button. If the layer is sloped, you can also specify whether the layer is **Drained**. If there are drainage pipes or sumps installed around the periphery of the landfill, then a sloped layer will function as a lateral drainage layer. The **Drained** feature is a new feature offered in UnSat Suite.

You can edit all of the parameter values in the **Value** column. The **Top Elevation** or **Bottom Elevation** will become active depending on **fixed top** or **bottom** condition is applied to profile. To edit any of the available parameters, click in the **Value** box and type your changes. Click [OK] to save your changes.

You can edit the unit of thickness from this dialogue box as well. To change the units of thickness, click in the right box of the **Thickness** row, and click the new units from the list. You cannot edit any other units in this way.

Note: When you edit the thickness of a layer, the profile will differentiate, depending on whether the top or bottom elevation was fixed in the Profile Properties dialogue box.

Note: If you change the slope of a layer, it will change the bottom slope of the upper layer and the top slope of the lower layer.

Layer Parameters Tab

Under the **Layer** tab, you can edit the properties of the material that comprises the layer.

To edit the values of these properties, click in the boxes and type your changes. Edit comments in the same way. The changes will be saved when you click [OK].

Edit the units of the parameters by clicking in the boxes and clicking the new units from the list.

Resizing Layers

Resizable Layers

Layers built from soil and waste, including Vertical Percolation layers, Lateral Drainage layers, or Barrier Soil layers, are resizable.

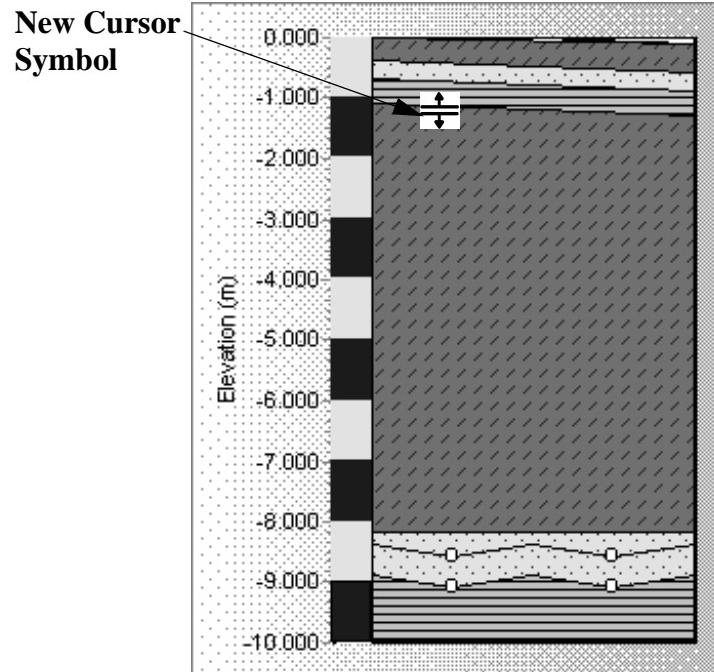
Non-Resizable Layers

Layers built from industry-produced materials with fixed thickness, including Geomembrane Liner, Geotextile and Geonet categories, are not resizable.

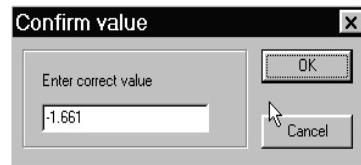
Note: The thickness of non-resizable layers can only be changed by editing the layer parameters. The thickness of resizable layers can be changed graphically or by editing layer parameters.

To resize layers:

- 1) Move the mouse arrow between two layers. The cursor symbol will change:



- 2) Click and drag the boundary to its new location.
- 3) Type the correct elevation of the boundary in the **Confirm Value** dialogue box.



Note: This option changes the thickness of both layers; above and below the boundary. Other thicknesses are unchanged. The result will not change the top or bottom elevation of the profile.

Inserting Layers

There are two methods of inserting a layer. You can insert a layer in the **Project Tree** or in the **Profile View**. In both cases, the new layer will be added above the layer that you click.

To insert a layer in the Project Tree:

- 1) <right click> on the layer in the Project Tree, above which you want to insert the new layer.
- 2)  **Insert Layer.**

- 3) Specify the layer properties.
- 4)  **[OK].**

To Insert a layer in the Profile View:

- 1) <right click> on the layer in the Profile View, above which you want to insert the new layer.
- 2)  **Insert Layer.**

- 3) Specify the layer properties.
- 4)  **[OK].**

An empty **Profile Material Properties** dialogue box will appear. Specify the material to insert.

Note: The new layer will always be inserted above the current layer.

Note: To insert the new layer inside the existing layer, split the layer first with the split function.

Note: VHELP can accommodate a maximum of 20 layers in a profile.

Material Category	Click a category from the list. The list will be limited to those categories that are allowed at the insertion point.
Material (Texture #)	Click the appropriate material from the list. Each layer category is associated with several textures that can be used in the design of the layer. The texture number in UnSat Suite corresponds directly to the texture number in the original DOS HELP.
	As far as material is selected, the top and bottom settings are activated.
Name	Type a unique name for material in this text box.
Description	Edit the comments in this column.
No Slope	If the No Slope button is clicked, the layer will be inserted with zero slope.
Slope	If the Slope button is clicked, you must specify Slope and Slope length . Slope must be expressed in percent. The slope length is measured in the cross-sectional view. It is recommended that you set up your profile so that the slope seen in the cross-section is equal to your maximum gradient in the sub-area of your landfill.
Drained	Only Lateral Drainage layers and Geotextiles and Geonets can be drained. You can edit Leachate Recirculation and To layer... , which allows you to decide if the leachate will recirculate through the landfill, and, if so, to which layer it will recirculate.
Drainage	Only Lateral Drainage layers and Geotextiles and Geonets can be drained. You can edit the Drainage Slope , Drainage Length , Leachate Circulation , and To layer... , which allows you to decide if the leachate will recirculate through the landfill, and, if so, to which layer it will recirculate. Drainage Length refers to the spacing between drainage pipes.
Top/Bottom Elevation	Type the top or bottom (the one allowed for the profile) elevation of the layer in this box.

Layer Parameters Tab

Click the Specific tab to edit the properties of the layer material:

Parameter	Displayed in this column, are the parameters of the material.
Value	Edit the values of the parameters in this column.
Units	You can change the units in this column.
Comment	Edit the comments about the parameters in this column.

Deleting Layers

There are three ways to delete a layer. You can delete a layer from the **Profile View**, **Project Tree View**, or the **Tool bar**.

To delete a layer from the Profile View:

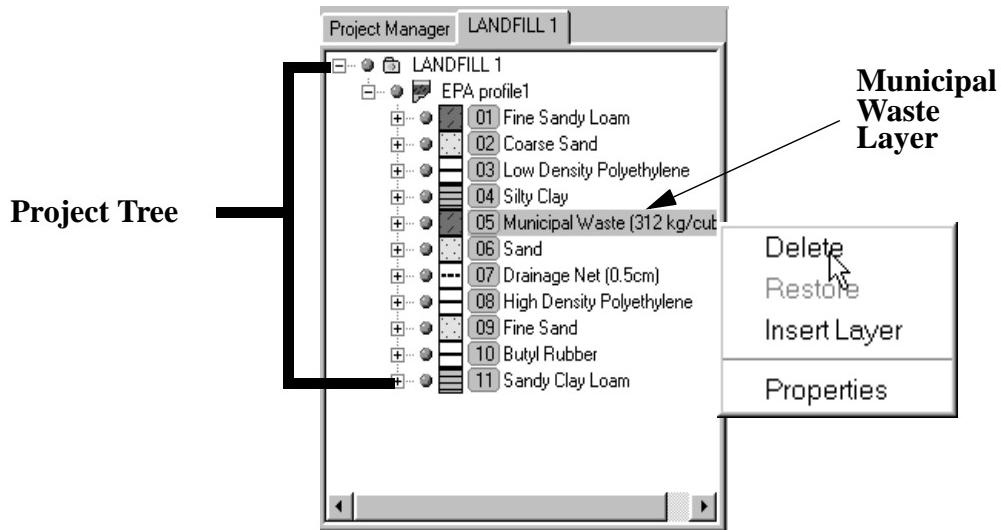
- 1) <right click> the layer.



- 2) ⌘ Delete.

To delete a layer from the Project Tree:

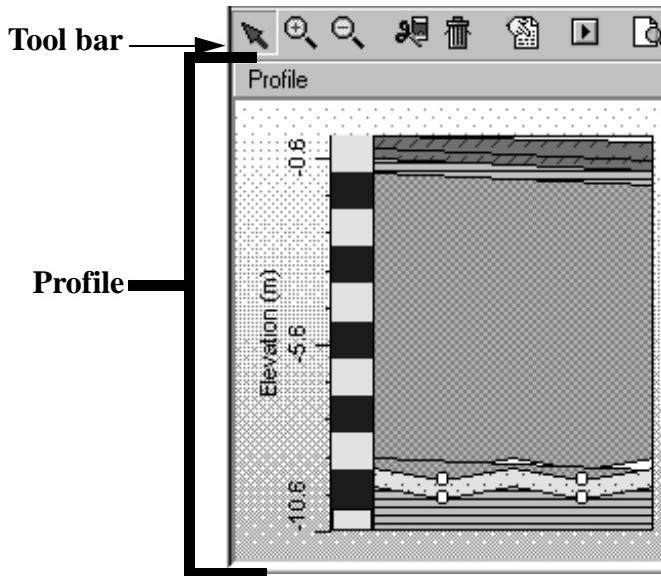
- 1) <right click> the layer.



- 2) Delete

To delete a layer from the tool bar:

- 1)  the layer in the Profile View (the layer will be highlighted).



- 2)  on the toolbar.

Note: UnSat Suite allows only certain materials in the top layer. If deleting a layer would result in an inappropriate top layer, a warning will appear, and the layer will not be deleted.

Restoring Layers

If you have deleted one or more layers, the garbage can icon on the toolbar will appear to bulge.

At this time, you can restore one or more of the deleted layers.  , and  **Restore All** or click a layer from the list.

Splitting Layers

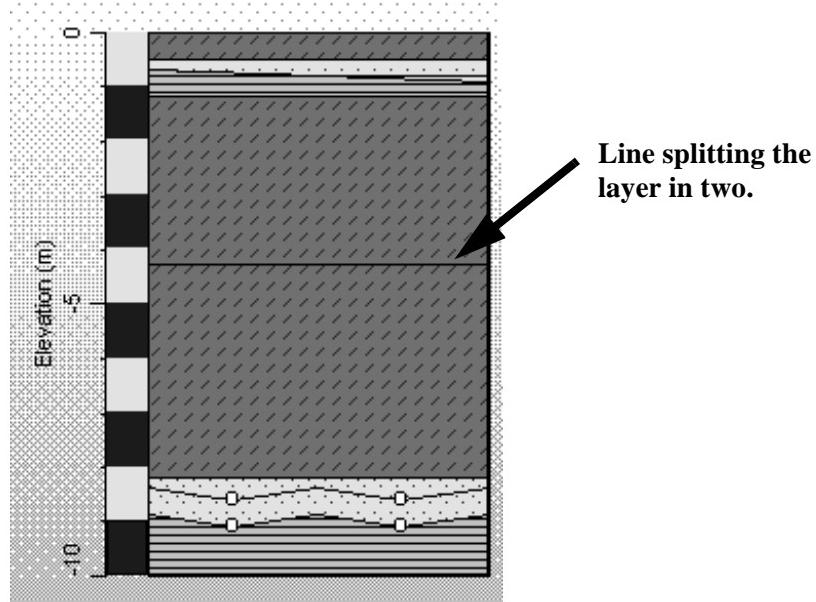
You can split a layer up into multiple sections and assign each section of the layer different values for each parameter.

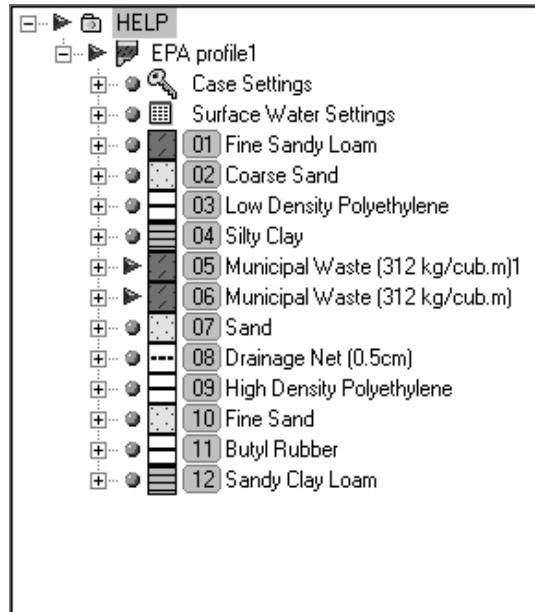
To split a layer:

- 1) <right click> on the layer in the Profile View.
- 2)  **Layer/Split.**

A line will appear through the layer and the layer will be split into two sections in the Project Tree.

Now the layer can be edited as two separate layers with unique properties in the Profile View and the Project Tree View.





The two
distinct
sections
of the
layer.

You can substitute material of each section with the following method:

To change the properties of a layer:

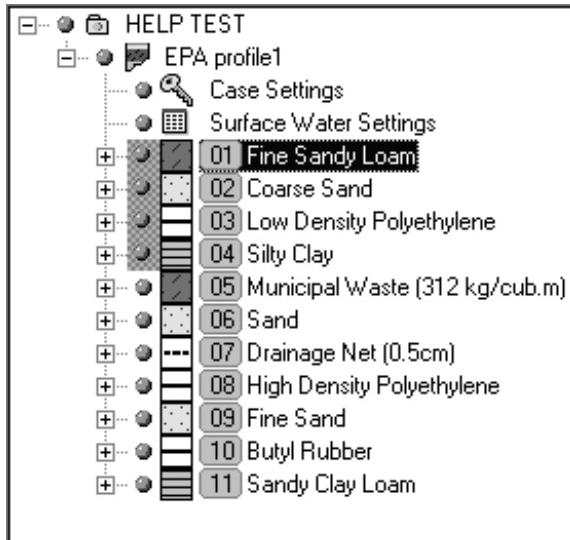
- 1) <right click> on the layer.
- 2) Click **Layer/Properties**.
- 3) Click a new material from the **Material Category** list. The appropriate **Material (texture#)** will appear.
- 4) Give the new layer a unique name and write a descriptive comment.
- 5) Enter the new layer thickness and name in the appropriate box.
- 6) Click the *[Help]... **Layer Parameters** tab. Edit the parameters.
- 7) Click **[OK]**.

Layer Groups

In landfill design, groups of layers can perform a specific function such as landfill capping or leachate collection and removal. UnSat Suite contains a special feature to set the properties of such layer groups.

To select a layer group:

- 1) Move the cursor arrow onto the Project Tree, for example, to the left of the **Fine Sandy Loam** icon (as shown below):



- 2) Click and drag a rectangle around the layers in the group. The layers will be highlighted in the Project Tree and in the Profile View.

To edit the layer group geometry or to delete selected layers:

- 1) Move the mouse onto the highlighted area in the Project Tree and click the right mouse button.
- 2) **Delete Selected Layers** to delete the group.

Note: *UnSat Suite allows only certain materials for the top layer. A warning will appear and the group will not be deleted if the result is an inappropriate top layer.*

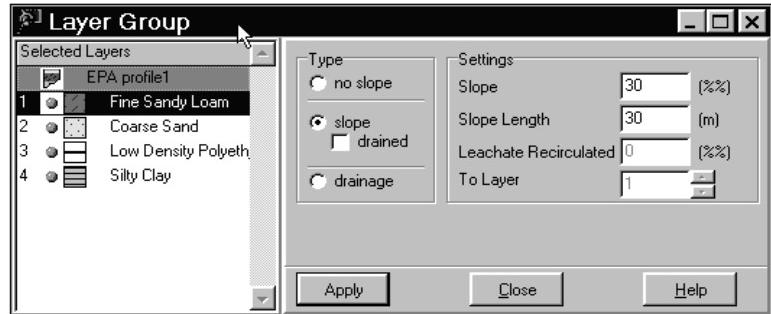
OR

To edit the layer group geometry or to delete selected layers:

- 1)  **Layer Group Geometry** to edit the group geometry.



The **Layer Group** dialogue box will appear:



On the left side of the dialogue box, you can see all the layers you have selected. The right side of the dialogue box is used for setting the layer group's properties. Depending on which buttons you have selected (no slope, slope or drainage) the boxes will become available and unavailable accordingly.

- 2) Click a button and edit the **Settings** boxes.

- 3)  **[Apply]**.

- 4)  **[Close]**.

Your new settings will now be used for your calculations.

Note: If the top layer of the profile was included in the layer group and the slope settings have changed, the Surface Runoff settings will change accordingly.

6

Generating Weather Data

Introduction

HELP requires three different types of meteorological data that must be provided as daily values:

- Precipitation (rain or snow),
- Solar radiation, and
- Mean air temperature.

In addition, HELP requires a set of parameters to simulate evapotranspiration that are constants for the duration of the simulation.

HELP will then use this data to:

- Calculate the volume of water flowing into the landfill, and simulate surface runoff, evaporation, vegetation growth and transpiration, and infiltration during warm periods; and
- simulate surface storage, snowmelt, runoff and infiltration during cold periods.

The daily data can be imported from a weather data file, for a particular meteorological station, or synthetically generated from the Weather Generator.

For synthetic generation of daily values of precipitation, mean temperature, and solar radiation DOS HELP and Visual HELP version 1 included a Weather Generator developed by the Agricultural Research Service of the USDA (U.S. Department of Agriculture), as well as parameters for generating synthetic data for 139 U.S. cities. These parameters for local probability processes were assessed using results of at least 20-year precipitation and 10-year temperature and radiation observations. Daily precipitation is generated using a model that is based on the Markov Chain model and the Two-parameter Gamma distribution model. The occurrence of rain on a given day has a major influence on the values of temperature and solar radiation for the day. The Weather Generator generates precipitation independently of the other variables and then generates temperature and solar radiation according to whether the day is wet or dry.

World Weather Generator Database

After Visual HELP 1.01 was released, WHI got a lot of requests from our clients worldwide to expand the area of Weather Generator application to other regions of the world.

Trying to meet these requests, WHI has developed a global database that includes more than 3000 stations and a GIS feature for searching the nearest stations globally. As a source of raw weather data the NOAA (National Oceanic and Atmospheric Administration) GDS (Global Daily Summary) database, which contains 14 years of daily precipitation and temperature data (1977-1991) for nearly 10,000 stations across the world, was used. It took a great deal of research and programming to decode the database files and develop filters to delete records with missing data. Finally, selected good records were mathematically processed and coefficients for weather generation at more than 3000 locations were estimated. These coefficients were stored in regional databases for Australia, Africa, Europe, Asia, North America, South America, and the former Soviet Union.

To verify the reliability of obtained coefficients, two series of tests were performed. In the first series of tests the generated daily precipitation and temperature were compared to observe six U.S.cities for which Dr.Clarence Richardson, the developer of the original Weather Generator database, performed his comparison in 1984. Results of the tests proved that errors between statistics for generated and observed data were in the range or less than when Dr. Richardson performed his tests using another set of observed data and coefficients obtained from it.

During the second series of tests, synthetic weather data was generated, using new weather coefficient databases, were compared with observed weather data for twelve sites around the world. The analysis revealed that the new weather databases could be successfully used to synthesize reliable weather data. Mean monthly temperature was consistently accurate within 3 degrees Fahrenheit. Mean monthly precipitation was consistently accurate within 0.3 inches.

For successful use of the HELP model, solar radiation coefficients and evapotranspiration parameters are required. This data was not available from the NOAA GDS disk. To fill this information gap, parameters were extracted from different sources (The World Survey of Climatology, U.N. Food and Agriculture Organization Agroclimatological Data Series) or extrapolated from the US territory using the Koppen world climate zoning scheme for determining regions with similar climate. The values of evapotranspiration parameters were specified for all weather stations stored in the global Weather Generator database.

If your landfill site is not located in our database, we recommend that you choose the closest city to your site and use the generated data for that

location for your simulations. UnSat Suite will search for the closest location of a given set of co-ordinates. This co-ordinate set can be entered as actual values or interactively with a map.

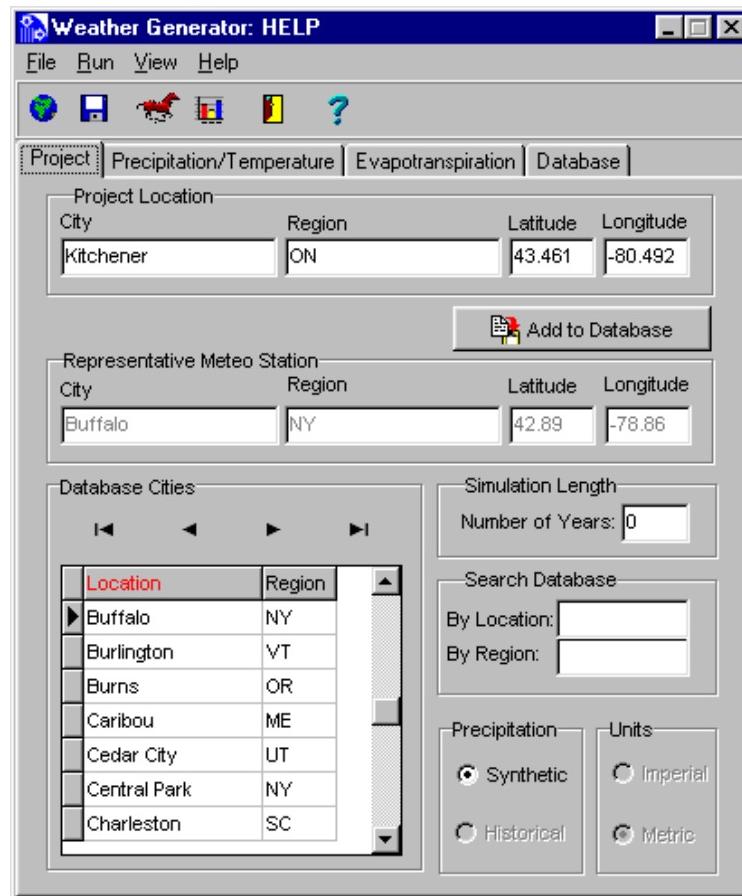
If you are not simulating a landfill found in the database, you can easily import your own set of real weather data. You must modify the format of your data so that it meets the standards of UnSat Suite. The required format is described further in this chapter. If you are in Canada, you can automatically import data in the format of the Canadian Climate Centre. In addition, our customers from the U.S.A. may automatically import data in the NOAA format. UnSat Suite checks NOAA files for missing daily and monthly records and informs you about the times, for which data are missing to make the correction process easy.

Starting the Weather Generator



To start the Weather Generator, click the icon from the Operational Tool bar, or from the **Run** menu click **Weather Generator**.

After an introductory splash screen is displayed, the Weather Generator dialogue box will appear:



In this dialogue box, you can input and edit data that will be used to generate the daily weather values. You can also view and edit the Weather Generator database, and import weather files in several common formats.

Getting Around

Input Features

The top menu bar contains the following menu options:

- | | |
|-------------|---|
| File | Import weather data files in different formats, open GIS, or save generated files and exit. |
| Run | Run the Weather Generator for selected location and weather parameters. |
| View | View synthesized data files. |
| Help | Access the Help dialogue box and general information on the Weather Generator. |

Tool Bar Buttons

The following icons provide you with quick access to functions dealing with weather data.



- Select from Map** Click this icon to activate GIS, click your project site location and find nearest location for which weather generating is possible.



- Save to file** Click this icon to save synthesized weather data.



- Run** Click this icon to run the Weather Generator.



- Output** Click this icon to view generated weather data.



- Quit** Click this icon to quit the Weather Generator.



- Help** Click this icon to get help from the Weather Generator.

Project Tab

Under this tab, you can search for the cities for which weather generation is possible, click the source of precipitation data (synthetic or default) for

several cities, and input the number of years to be modeled. You can also input a new record to the Weather Generator database.

Precipitation/Temperature Tab

Under this tab, you can edit the mean monthly precipitations and temperatures for the selected database city to adjust the values for your project site. Monthly precipitations and temperatures are the input parameters for weather generation.

Note: If you edit parameters here, your changes will be effective for only one run of Weather Generator. If you wish your changes to become permanent, add the edited weather station to the database with a new name. See “Adding a Record to the Weather Generator Database” on page 121 for details.

Evapotranspiration Tab

Under this tab, you can edit default evapotranspiration parameters for the selected database city to adjust the values to your project site. These values are not the input for the Weather Generator but they are used by the HELP model for calculating daily evaporation from leaves, transpiration and evaporation from soil.

Database Tab

Under this tab, you can view and edit the Weather Generator database containing parameters for synthetic weather generation.

Selecting the Nearest Location to Generate Weather Data

There are over 7000 locations for synthetic weather generation data which are stored in the regional databases. Within the project you may work with only one regional database. The weather stations are listed, in alphabetical order, in the listbox in the bottom left corner of the Weather Generator dialog box.

With the original HELP database (**US and Canada**), synthetic weather data can be generated for some of the locations, while others have actual historical data associated with them. There are a few locations for which the Weather Generator can generate synthetic data that also have historical data.

With the **US and Canada** database the Weather Generator will generate synthetic data (precipitation, temperature, and solar radiation) for 139 U.S. cities. When parameters for the synthetic generation are available, the **Synthetic** option button will be selected.

This list also contains names of 102 U.S. cities for which the database contains five years of historical precipitation data. You can use the historical precipitation data to simulate five years of temperature and

solar radiation. When only five years of historical precipitation data are available, the **Historical** option button will be selected.

For some cities, both options are possible. In this case, both the **Synthetic** and **Default** option buttons will be available and you can choose either to synthesize up to 100 years of precipitation data using Weather Generator or to use default historical data.

There are two ways to select a location that is in the database. You can do a text search for the location, or you can use the GIS feature.

Text Search

Text searches allow you to search the database by location or by region. If you know the name of location, search the database by city.

To search the database by location, click in the **By Location** box in the **Search Database** frame and type the first two letters of the location name. The indicator in the **Location** column will show the location that matches your selection.

You may, however, want to see all database locations for the region (country) where your project site is located. In this case, search the database by region.

To search the database by region, click in the **By Region** box and type the first two letters of the region's abbreviated name. All locations from the region will be displayed in alphabetical order.

The following is a list of the regions and their database abbreviations. The list is alphabetical for each map. There are eight maps including: U.S.A. and Canada, Africa, Asia, Australia, Europe, North America, South America, and the Former USSR.

The U.S.A. and Canada map contains only stations in the United States. The stations are abbreviated by state.

	<u>Abbreviation</u>	<u>Region (State)</u>
<i>U.S.A and Canada</i>	AK	Alaska
	AL	Alabama
	AR	Arkansas
	AZ	Arizona
	CA	California
	CO	Colorado
	CT	Connecticut
	DC	District of Columbia
	DE	Delaware
	FL	Florida
	GA	Georgia
	HI	Hawaii
	IA	Iowa
	ID	Idaho

IL	Illinois
IN	Indiana
KS	Kansas
KY	Kentucky
LA	Louisiana
MA	Massachusetts
MD	Maryland
ME	Maine
MI	Michigan
MN	Minnesota
MO	Missouri
MS	Mississippi
MT	Montana
NC	North Carolina
ND	North Dakota
NE	Nebraska
NH	New Hampshire
NJ	New Jersey
NM	New Mexico
NV	Nevada
NY	New York
OH	Ohio
OK	Oklahoma
OR	Oregon
PA	Pennsylvania
RI	Rhode Island
SC	South Carolina
SD	South Dakota
TN	Tennessee
TX	Texas
UT	Utah
VA	Virginia
VT	Vermont
WA	Washington
WI	Wisconsin
WV	West Virginia
WY	Wyoming

The remaining maps are separated into seven world regions including: Africa, Asia, Australia, Europe, North America, South America, and the former Soviet Union. The stations are abbreviated by countries. Please note that some stations in different world regions share the same country abbreviation. Therefore, note which world region you are in before selecting a country. These discrepancies are outlined at the end of the table.

	<u>Abbreviation</u>	<u>Region (Country)</u>
<i>Africa</i>	ALGE	Algeria

	ATLA	Atlantic Ocean Islands
	BENI	Benin
	BURK	Burkina Faso
	CANA	Canary Islands/W Sahara
	CAPE	Cape Verde
	COMO	Comoros
	KENY	Kenya
	MALI	Mali
	MAUR	Mauritania
	MORO	Morocco
	NAMB	Namibia
	NIGE	Nigeria
	SENE	Senegal
	SOUT	South Africa
	TUNI	Tunisia
	ZIMB	Zimbabwe
	CARO	Caroline Islands
	CHIN	China
	COOK	Cook Islands
	DETA	Detached Islands
	FREN	French Polynesia
	HONG	Hong Kong
	INDI	India/w Indian Ocean Islands
	JAPA	Japan
	KIRI	Kiribati
	KORE	North/South Korea
	MALA	Malaysia
	MARI	Marianna Islands
	MARS	Marshall Islands
	PAKI	Pakistan
	PHIL	Philippines
	SAMO	Samoa
	SYRI	Syria
	TAIW	Taiwan
	THAI	Thailand
	TURK	Turkey
	TUVA	Tuvalu
	VANU	Vanuatu
	VIET	Vietnam
Australia	AUST	Australia
Europe	AUST	Austria
	AZOR	Azores
	BELG	Belgium
	BULG	Bulgaria
	CYPR	Cyprus
	CZEC	Czechoslovakia
	DENM	Denmark

	ESTO	Estonia
	FARO	Faroe Islands
	FRAN	France
	FORM	Former Yugoslavia
	GERM	Germany
	GREE	Greece
	HUNG	Hungary
	ICEL	Iceland
	IREL	Ireland
	ITAL	Italy
	LATV	Latvia
	LITH	Lithuania
	MADE	Madeira
	NETH	Netherlands
	NORW	Norway
	POLA	Poland
	PORT	Portugal
	ROMA	Romania
	SPAI	Spain
	SWED	Sweden
	SWIT	Switzerland
	UNIT	United Kingdom
<i>North America</i>	CANA	Canada
	GREE	Greenland
	MEXI	Mexico
	UNIT	United States
<i>South America</i>	ARGE	Argentina
	BOLI	Bolivia
	BRAZ	Brazil
	CARI	Caribbean Islands
	CHIL	Chile
	CUBA	Cuba
	JAMA	Jamaica
	PANA	Panama
	PARA	Paraguay
	PERU	Peru
	PUER	Puerto Rico
	URUG	Uruguay
<i>Former USSR</i>	FORM	Former Soviet Union

The following table outlines the repetitive discrepancies in the abbreviations of world regions and corresponding country:

Abbreviation	Region and Country 1	Region and Country 2
AUST	Australia, Australia	Austria, Europe
CANA	Canary Islands/w Sahara, Africa	Canada, North America

FORM	Former Yugoslavia, Europe	Former Soviet Union, Former USSR
------	---------------------------	----------------------------------

The GIS Map



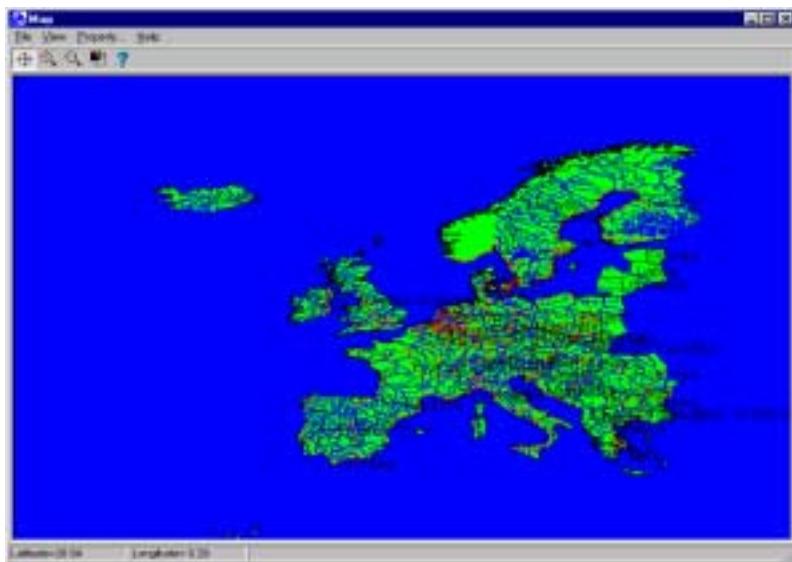
The second way to select a city is to use the GIS map. This is especially useful if you do not know which neighbouring locations can be used to generate weather data.

To open the GIS map:

From the **File** menu click **Map**, or click



For example, that your project site is located in the western outskirts of Wageningen, The Netherlands. The map of Europe will appear on your display, as shown below:



To zoom into the correct location in the GIS searcher:



- ☞ 1) Left click on an area north-west of your site,
- 2) drag a zoom rectangle around your site, and
- 3) release the left mouse button.

Repeat these steps, until you can see your site.

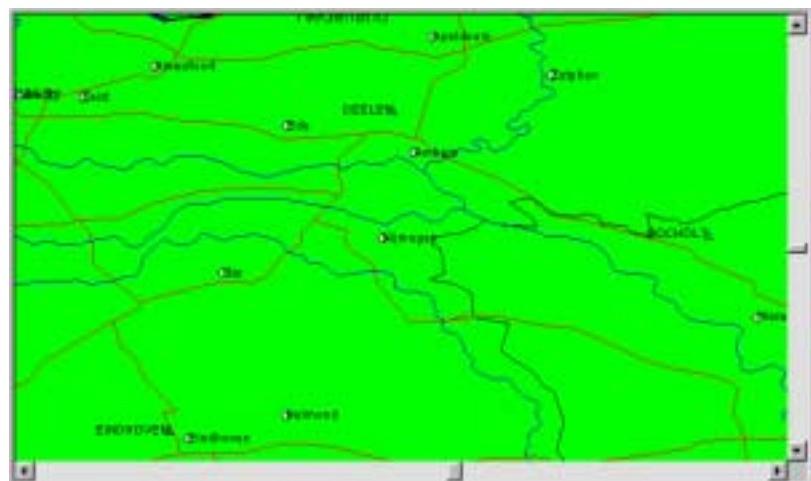
To select the correct location of your site in the GIS searcher:



☞ to activate the crosshairs.

☞ your site.

The window below illustrates what will appear if you zoom in on Wageningen, The Netherlands.



Move the crosshairs to the spot on the map where your site is located and select.

The search results will appear:



The **Select Nearest Meteo Station** dialogue box displays the latitude and longitude of the point that you clicked, as well as the closest cities that are in the database, and the distance from your site in kilometers. The closest weather station does not always best represent local weather conditions. Water bodies and mountain ridges can cause a city that is farther away to be more representative. Using your knowledge of local geography, select the most representative city. Parameters for this city will appear in the boxes throughout the Weather Generator.

Note: For this GIS searcher, a longitudinal value that is negative indicates west, while a positive value indicates east. Similarly, a negative value for latitude is south, while a positive value is north.

After clicking the nearby database city, two sets of co-ordinates will appear under the **Project** tab of the Weather Generator.

Under **Project Location**, you will see the co-ordinates of your project site.

Under **Representative Meteo Station**, you will see the co-ordinates of the most representative meteo station, which was selected from the database:

:

Project	Precipitation/Temperature	Evapotranspiration	Database														
Project Location																	
City	Region	Latitude	Longitude														
Wageningen	NETH	51.99	5.83														
Add to Database																	
Representative Meteo Station																	
City	Region	Latitude	Longitude														
DEELEN	NETH	52.07	5.88														
Database Cities																	
◀◀ ▶▶ ▶ ▶																	
<table border="1" style="width: 100%; border-collapse: collapse;"> <thead> <tr> <th style="color: red;">Location</th> <th>Region</th> </tr> </thead> <tbody> <tr><td>DAUGAVPILS, LATV</td><td>LATV</td></tr> <tr><td>DE BILT</td><td>NETH</td></tr> <tr><td>DEAUVILLE</td><td>FRAN</td></tr> <tr><td>DEBRECEN</td><td>HUNG</td></tr> <tr style="background-color: #0000ff; color: white;"> <td>DEELEN</td><td>NETH</td> </tr> <tr><td>DEN HELDER</td><td>NETH</td></tr> <tr><td>DEUSELbach</td><td>GERM</td></tr> </tbody> </table>		Location	Region	DAUGAVPILS, LATV	LATV	DE BILT	NETH	DEAUVILLE	FRAN	DEBRECEN	HUNG	DEELEN	NETH	DEN HELDER	NETH	DEUSELbach	GERM
Location	Region																
DAUGAVPILS, LATV	LATV																
DE BILT	NETH																
DEAUVILLE	FRAN																
DEBRECEN	HUNG																
DEELEN	NETH																
DEN HELDER	NETH																
DEUSELbach	GERM																
▲ ▼																	
Simulation Length																	
Number of Years: <input type="text" value="0"/>																	
Search Database																	
By Location: <input type="text"/>																	
By Region: <input type="text"/>																	
Precipitation:																	
<input checked="" type="radio"/> Synthetic	<input type="radio"/> Imperial																
<input type="radio"/> Historical	<input checked="" type="radio"/> Metric																
Units																	

Now you can generate weather data for your project.

During the calculation, the Weather Generator will use all of the parameters of the selected database weather station except the latitude and longitude. The latitude and longitude of your project site will be used. This correction allows you to generate more accurate values of solar radiation. However, you may want to edit the input coefficients to suit them to your local conditions.

Precipitation and Temperature

UnSat Suite allows you to edit commonly available data which is used as input parameters for weather generation.

To edit the monthly average precipitation and temperature values used for synthetic weather generation, click the **Precipitation/Temperature** tab:

Precipitation-(mm)		Temperature-(°C)	
	User	Default	User
January	90.5		1.55
February	49.3		1.59
March	82.4		5.58
April	42.1		8.01
May	59.6		12.33
June	85.2		14.83
July	65.1		16.98
August	55.9		16.98
September	53.5		14.27
October	74.1		10.80
November	75.9		5.87
December	85.9		3.48

under **Default**.

The default values will be copied to the **User** column, where they can be edited.

Click the **[Clear]** button to erase the values in the **User** column.

Values in the **User** column will be used for calculations. If a value is missing from the **User** column, the default values will be used.

Note: If you edit parameters here, your changes will be effective for only one run of Weather Generator. If you wish your changes become permanent, add the edited weather station to the database with a new name.

Evapotranspiration

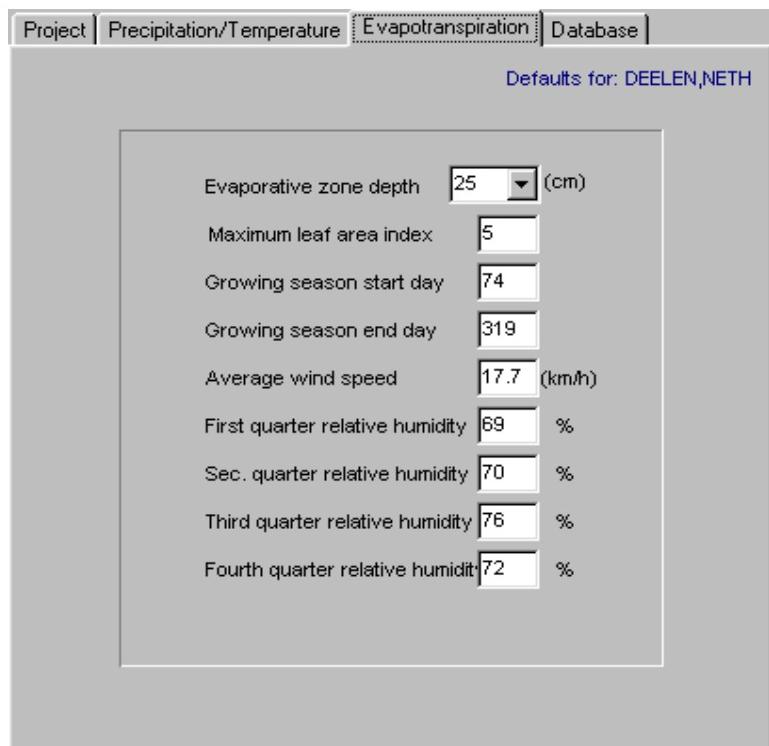
The HELP model uses a complicated multilevel procedure to calculate different types of evaporation and evapotranspiration. The subroutines of this model allow calculation of evaporation from snow, soil, and leaves. In addition, the model calculates vegetation growth and transpiration. In total, around 70 equations describe these processes. Fortunately, the

number of parameters which require the user's input are much less than the number of equations.

These parameters are:

- Evaporative zone depth
- Maximum leaf area index
- Growing season start and end day
- Average wind speed
- Quarterly relative humidity

To see and edit the parameters used for simulating evaporation from soil and leaves and plant transpiration, click the **Evapotranspiration** tab.



Edit the parameters accordingly.

Evaporative zone depth

This is the maximum depth from which the water can be removed by evapotranspiration. The program does not allow the evaporative zone depth to exceed the depth to the uppermost geomembrane liner or a barrier soil layer. You can find three values of this parameter in the **Evaporative zone depth** box for the specified location. These values are characteristic for grassy vegetation on a thick layer of loamy soil. Three values correspond, in growing order, to *bare soil, fair and excellent stand of grass*. The evaporative zone depth can be 2-3 times more shallow for sandy soil and 2-5 times deeper for clay soil.

Maximum leaf area index

The leaf area index is the ratio of the leaf area of actively transpiring vegetation to the surface area on which the vegetation is growing. UnSat Suite provides you with the value typical of the selected location. The maximum value for bare soil is 0. For a *poor stand of grass* the typical value is 1; for a *fair stand of grass*, 2; for a *good stand of grass*, 3.5; and for an *excellent stand of grass*, 5.

Growing season start and end days

The start and end of the growing season are determined, generally, by air temperature. In North America the growing season starts when the mean daily temperature rises above 10-12 °C (50 - 55 °F).

Average wind speed

This is the average annual wind speed.

Quarterly relative humidity

These are the average quarterly values of relative humidity.

Generating Weather Data

The majority of the database locations allow you to synthetically generate precipitation. In this case, you must specify the number of years to be simulated. If the database city you have selected has historical precipitation data, and you have chosen to use this data, then the simulation will run for five years, by default. For these cities, the number of years for the simulation of temperature and solar radiation data will also be restricted to five years.

To specify the number of years to be simulated, click in the **Number of Years** box under the **Project** tab and type the number of years (maximum 100).

To run the Weather Generator:

☞  on the Weather Generator tool bar.

Or

☞ **Run\All.**

When the computations are complete,

☞  to save the weather data.

Viewing Generated Weather Data

To view the results of meteorological simulation:

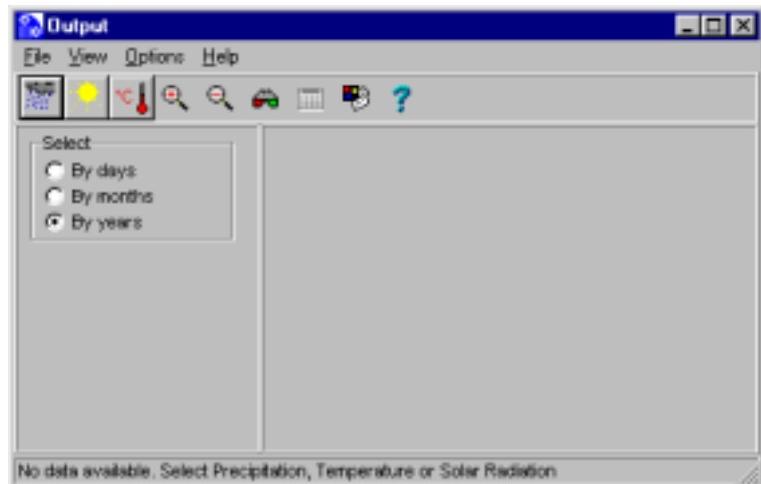
☞  on the tool bar

Or

☞ **View.**

Click the appropriate option from the **Output** dialogue box that appears.

The **Output** window will appear:



Menu

The following menu items can be used in the **Output** window:

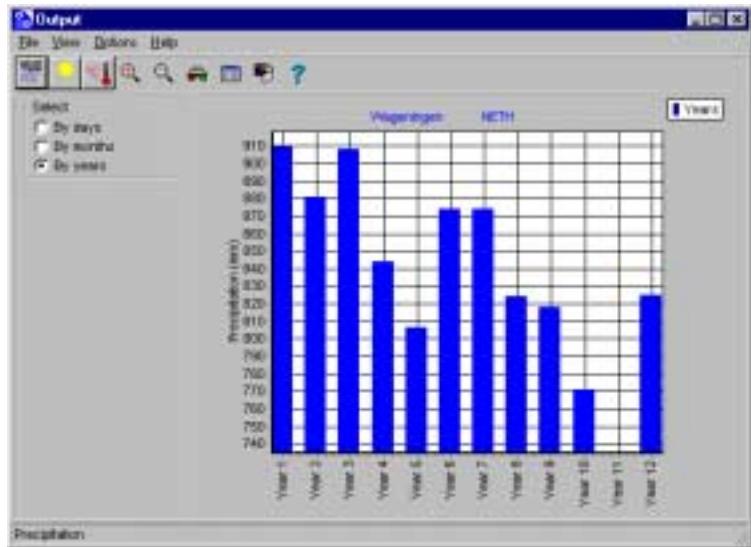
File	Save and print generated data, and exit the Output dialogue box.
View	View different types of generated data, activate and deactivate the zooming feature.
Options	Switch between different time scales and activate the data editor (this function is available only for daily data).

Tool Bar Buttons

	Precipitation	Use this icon to view precipitation data.
	Solar Radiation	Use this icon to view solar radiation data.
	Temperature	Use this icon to view temperature data.
	Zoom In	Use this icon to zoom in on the graph.
	Zoom Out	Use this icon to zoom out of the graph.
	3D-View	Use this icon to view graph in 3D (recommended if the number of simulated years is less than five).
	View/Edit Data	Use this icon to view and edit generated weather data (this option is available only for daily values).
	Graph Properties	Use this icon to change graph properties.
	Help	Use this button to open the Help dialogue box.

In the **Select** frame, below the tool bar, you can choose the time scale that you would like to view the generated data with. The **By Years** button is the default.

To view a specific graph, click the type of weather variable from the tool bar or from the **View** menu and click the desired button in the **Select** frame or from the **Options** menu.



Note: The y-axis on the graph is scaled between maximum and minimum values.

If the length of the simulated period exceeds five years and the **By days** or **By months** time scale is selected, values for the first five years will be graphed by default.

For each day or month, the data for displayed years will be portrayed in ascending order from left to right. Values for different years will be coloured in different colors, the correspondence between colors and years can be seen in the upper right corner of the dialogue box.

If you choose to graph **By days** or **By month**, you may specify the years you want to plot in the list, and the format you wish to use in the **Selection Type** frame.

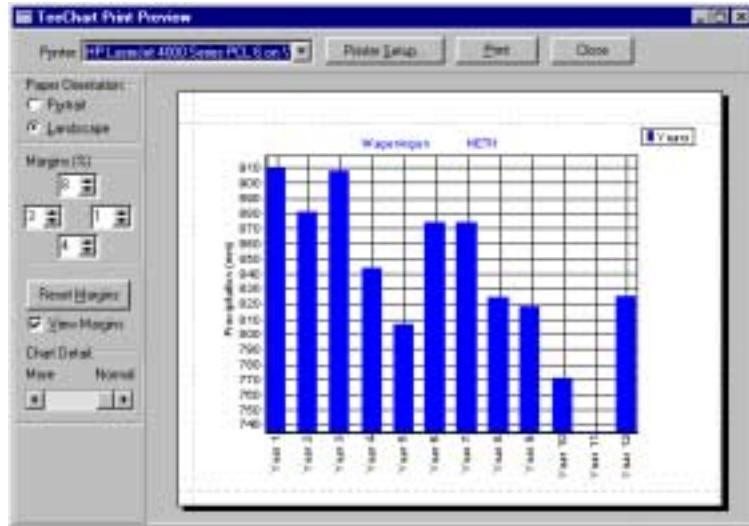
Your first option is **Custom**, and click the relevant years from the list.

The **Group of N elements** button allows you to plot data corresponding to any number of consecutive years. Specify the number of years in the **Number of Years** spin box, and click [**Select Next Group**] to cycle through your data.

Click the **Each N-th** button to graph data for the first year, and years (1+N), (1+2N), (1+3N), etc. Specify the number in the **Each N-th** box.

Click the **Select All** button to plot data for all years. Click the **Unselect All** button to clear your graph.

If you click **Print** from the **File** menu, the **TeeChart Print Preview** dialogue box will appear:



From this dialogue box you can set up printing characteristics and print the graph.

You can also save a graph as a bitmap by clicking **Save As** from the **File** menu.

Editing Generated Data

You may wish to see generated weather data in table format.

To view the generated weather data in table format:



Or

from the **Options** menu **View/Edit**.

The following table will appear if you select the **By months** option in the **Output** dialogue box:

A screenshot of a Windows-style 'Data Edit' dialog box. The title bar says 'Data Edit'. The main area is a table with 'Precipitation' in the first column and 'Jan', 'Feb', 'Mar', 'Apr', 'May', 'Jun', 'Jul' in the subsequent columns. Rows represent 'Year 1' through 'Year 9'. The cell for Year 1, Jan contains '73.4'. The 'Edit' checkbox at the bottom left is unchecked. Buttons for 'OK' and 'Cancel' are at the bottom right.

Precipitation	Jan	Feb	Mar	Apr	May	Jun	Jul
Year 1	73.4	56.4	42.5	35.4	94.8	71.9	91.7
Year 2	77.9	101	71.2	55.9	33.9	84.2	39.7
Year 3	161.2	47.1	122.8	38	36.5	85.9	36.2
Year 4	60.2	58.5	40.9	18.1	34.2	129.1	78.7
Year 5	75.7	13.9	52.3	25	52.3	126.1	92.6
Year 6	119.8	44.2	75.2	37.4	57.6	77.2	81.4
Year 7	69.5	38	76.4	30.7	85.9	102.6	39.9
Year 8	111.9	37	92.4	60.7	34.5	78.4	77.6
Year 9	65.5	53.9	85.1	49.1	69.7	68.9	79.1

To find specific value in the hidden part of the table, use the scroll bars.

You cannot edit data generated with the monthly and annual time scales. However, you can edit data generated as daily values, which is particularly useful if you would like to test the landfill performance under the extreme conditions, e.g. 50 year storm. If you choose **By day**, the **Edit** option in the bottom of the table will become selected:

A screenshot of a 'Data Edit' dialog box showing daily precipitation values. The title bar says 'Data Edit'. The table has 'Precipitation' in the first column and days of the month (1, 2, 3, 4, 5, 6, 7) in the second through eighth columns. Rows represent dates from '1 Jan' to '1 Jul'. The cell for '1 Jan' contains '0'. The 'Edit' checkbox at the bottom left is checked. Buttons for 'OK' and 'Cancel' are at the bottom right.

Precipitation	1	2	3	4	5	6	7
1 Jan	0	0	0	0	0	2.1	1.7
1 Feb	2.5	2.8	5.4	2.8	0	0	1.1
1 Mar	0	2.5	0	0	0.1	2.3	2
1 Apr	0	0	2.7	0.7	5.1	7.1	0.1
1 May	0	0	0	0	0.7	0	6.1
1 Jun	11.1	3.8	0	2.1	0	1.7	2.1
1 Jul	0.5	1.3	1	0.6	0.4	4.3	0

Note: In the table of daily values each row represents a month of generated weather data.

Viewing the Weather Generator Database

Click the **Database** tab to see the Weather Generator database.

City	Region	Latitude [°]	Longitude [°]	Mean Tmax (°C)	Mean Tmin (°C)
DAUGAVPILS, LATV	LATV	55.87	26.62		
DE BILT	NETH	52.1	5.18		
DEAUVILLE	FRAN	49.37	0.17		
DEBRECEN	HUNG	47.48	21.63		
DEELEN	NETH	52.07	5.88		
DEN HELDER	NETH	52.92	4.78		
DEUSELbach	GERM	49.77	7.05		
DEVA	ROMA	45.88	22.9		
DIJON	FRAN	47.27	5.08		

The database contains all of the parameters necessary to synthetically generate daily precipitation, air temperature, and solar radiation. In addition to those parameters, the database contains a number of specific statistical parameters necessary for generating weather data.

The lower part of the Database tab contains the Unit Converter for converting between *customary* and *metric* unit systems.

To convert temperature from Fahrenheit to Celsius:

Click in the box to the right of the °F and type the number.



The converted value will appear in the box to the left of °C.

To convert temperature from Celsius to Fahrenheit:

Click in the box to the left of the °C and type the number.



The converted value will appear in the box to the right of °F.

The same procedure is used to convert the depth of precipitation (**mm** to **in**) and wind speed (**km/h** to **mph**).

Adding a Record to the Weather Generator Database

In addition to being able to get more information about the selected weather station, Visual HELP allows you to save your specific set of coefficients for future use by adding a new record to the database.

To add a new record to the database:

- 1) Click a representative meteo station from the database.
- 2) Use the button in each Tab to copy the default values from the representative station. Then, Edit the input coefficients for your project location under the **Project**, **Precipitation/ Temperature**, and **Evapotranspiration** tabs.
- 3) Click the **Projects** tab, and **Type** the city name in the **City** box.
- 4) **Type** the region abbreviation in the **Region** box.
- 5) [Add to Database]

A new record for your site will be added to the bottom of the list:

City	Region	Latitude [°]	Longitude [°]	Mean Tmax (°C)
ZARAGOZA/SANJU	SPAI	41.67	-1.02	
ZELTWEG	AUST	47.2	14.75	
ZILINA	CZEC	49.23	18.62	
ZINNWALD	GERM	50.73	13.75	
ZUGSPITZE	GERM	47.42	10.98	
ZUID-LIMBURG	NETH	50.92	5.77	
ZURICH	SWIT	47.38	8.57	
ZURICH	SWIT	47.48	8.53	
* Wageningen	NETH	51.99	5.83	

The new record will inherit all of the unchanged properties of the parent record.

K  to save the new record.

Editing the Weather Generator Database

UnSat Suite allows you to edit database records. Normally, editing the precipitation, temperature, and evapotranspiration values, as well as editing the generated data, gives you enough flexibility to create a representative set of input weather data. However, if you are an experienced user of the Richardson and Wright's Weather Generator, you may choose to edit the input parameters in the database.

Note: Editing of database records can permanently harm your data.

The database editor is inactive (the editor toolbar is greyed out) by default. It becomes available when you add a new record, or when you click the **Edit** check box.



- ☞  to insert a new line above the current record.
- ☞  to delete the current record.
- ☞  to save the current record.
- ☞  to cancel changes to the current record.

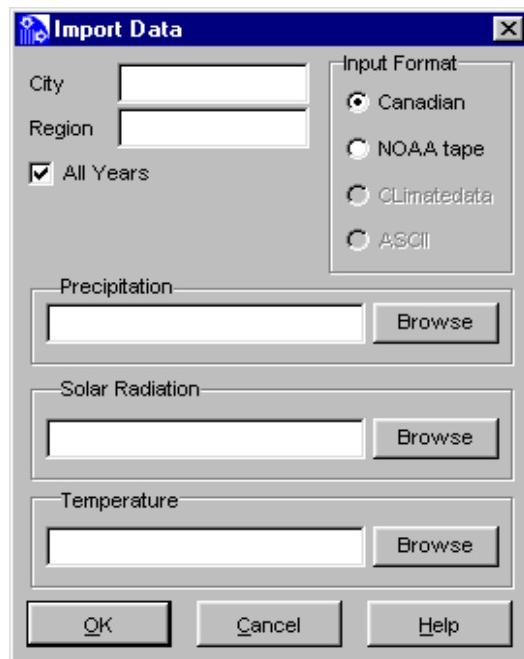
Once you have prepared the input data, you can run the Weather Generator.

Importing Weather Data in Canadian Climate Centre Format

In addition to generating weather data or using the default five year sets of precipitation for the selected years, UnSat Suite allows you to import files containing weather data in the format used by the Canadian Climate Centre. Sample files are provided with the installation and will appear in the **Weather Import** subfolder of your WHI UnSat Site/Visual HELP folder. This folder contains two years of Canadian weather data in proper format. The files are: **CanPrec.dat** for precipitation, **CanTemp.dat** for temperature, and **CanSrar.dat** for solar radiation. Please NOTE that if you want to import more than 2 years of data, we recommend that you split up your data into multiple files each containing 2 years, and import them separately.

To import data in this format:

From the **File** menu of Weather Generator click **Import**. The following dialogue box will appear:



Type the name of the location in the **City** box, and the abbreviated name of the region or province in the **Region** box.

In the **Input Format** frame, you can see that the **Canadian** option button is selected.

To import precipitation data from a file, type the file name in the **Precipitation** box, or click [**Browse**] to browse for the file. Use the same procedure to import solar radiation data, and temperature data. Click [**OK**].

If you wish to import data for a subset of years in your file, clear the **All Years** option. Type the start year for desired set of years in the **Start Year** box. Type the end year for desired set of years in the **End Year** box. Click **[OK]**.



To run UnSat Suite successfully, you also need to specify the evapotranspiration coefficients for your site. This can be done by using the evapotranspiration coefficients for one of the database weather stations and editing them. In the main dialogue box of Weather Generator, click the database city and proceed to the **Evapotranspiration** tab. Edit the evapotranspiration coefficients to suit your local conditions.

☞ in the **Weather Generator** to save changes.

☞ to close the **Weather Generator**.

Importing Weather Data in NOAA format

In addition to import weather files in Canadian Climate Centre format, you may check and import precipitation and temperature data in common NOAA format. Sample files NOAA_PRC.TXT, NOAA_TMAX.TXT and NOAA_TMIN.TXT are located in the **Weather Import** subfolder in your WHI UnSat Site/Visual HELP folder.

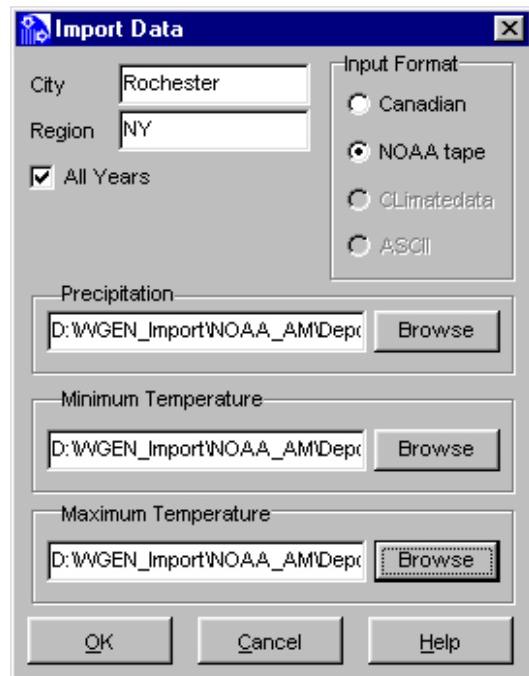
To import data in NOAA format:

From the **File** menu of Weather Generator click **Import**. Type the name of the location in the **City** box, and the abbreviated name of the region or province in the **Region** box.

In the **Input Format** frame, select **NOAA tape**.

To import precipitation data from a file, type the file name in the **Precipitation** box, or click **[Browse]** to browse for the file. Use the same

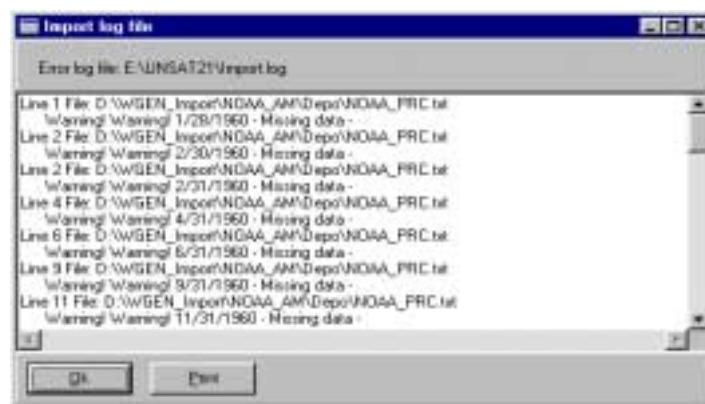
procedure to import **Minimum Temperature** and **Maximum Temperature** data.



Click [OK]. UnSat Suite/Visual HELP will start checking input files. The users who have been using NOAA files for a long time report multiple missing daily records and, sometimes, missing monthly records. UnSat Suite/Visual HELP will correct missing daily records the following way:

- missing daily precipitation records will be filled with 0 values,
- missing minimum or maximum temperature records will be filled with the average value for the previous and next day.

All corrections will be documented in the **Import log file**:



You may print this file and make your own corrections of missing daily records.

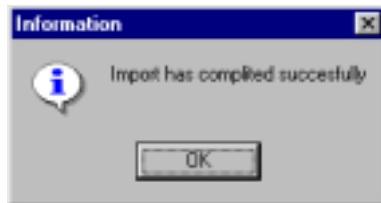
If the source files contain missing monthly records, UnSat Suite/Visual HELP will stop checking procedure and put a message at the end of the **Import log file**:

Line 56 File: D:\WGEN\Import\NOAA_AM\Depo\NOAA_PRC.txt
Error! Expected: 8/1964 Found: 9/1964 - Missing month data -

Missing monthly records have to be corrected or the whole year with a missing record has to be deleted by the user from the original file.

Note: If you are going to delete a year with a missing monthly record, to prevent a consistency of weather data, you have to delete corresponding annual records from the rest of the files. E.g., if year 1964 of the sample precipitation file contains a missing monthly record, all records for this year has to be deleted from the precipitation, minimum temperature and maximum temperature files.

After you deleted years with missing monthly records, run the import feature again. Scroll the precipitation **Import log file** to the end and if no missing monthly records were reported, click **OK**. The program will start checking for the minimum and maximum temperature files. Scroll the temperature **Import log file** to the end and if no missing monthly records were reported, click **OK**. The following message will inform you about results of the import procedure:



Click **OK**.

The recently imported Precipitation and Temperature files as well as the parameters for the weather station noted in the **Representative Meteo Station** text box will be used to generate a **Solar Radiation** file. If, by your opinion, this station does not represent your site adequately, select another station. The evapotranspiration coefficients for this station will be also used for the HELP simulation.

To generate a **Solar Radiation** file, under **Run** menu of the Weather Generator window select **Solar Radiation**.



in the **Weather Generator** to save changes.

 to close the **Weather Generator**.

Customizing Weather Data

There are two ways to customize weather data in UnSat Suite.

If you have your own daily data for precipitation, mean temperature, and solar radiation, you may convert them into the UnSat Suite format and run your model.

To locate your project in your machine:

- 1) **Run** the project with data generated for one of the available locations by **Weather Generator**.
- 2)  **Output**
- 3)  **Original Listing**

The standard listing used by the DOS version of the HELP model will open. Just below the title block will be a list of weather files. They will be described by drive, folder(s), and project name. They will end in one of the four following subscripts:

- _weather1.dat** - Daily Precipitation,
- _weather2.dat** - Mean Daily Temperature,
- _weather3.dat** - Daily Solar Radiation, and
- _weather4.dat** - Evapotranspiration Parameters.

You have to format your data the same way as these files are formatted and replace the existing files.

Formatting the first three files is very simple: 10 numbers in a row, 37 rows for a year. In summary, convert your data into this format using any data editor. Direct your files into the project directory and run UnSat Suite.

Here are parts of the Precipitation and Temperature files for Cheyenne, WY:

PRECIPITATION

2

1

Cheyenne WY

0.41	0.40	0.97	1.24	2.39	2.00	1.87	1.39	1.06	0.68	0.53	0.37
1	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	1
1	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.01	0.00	0.00	2
1	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.07	0.00	3
1	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	4
1	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.01	0.00	5

TEMPERATURE

2

1

Cheyenne WY

26.1	29.3	32.1	41.8	52.2	62.0	68.9	66.8	57.9	47.5	34.8	29.3
1	27.9	23.7	23.0	16.6	23.7	35.3	26.3	27.4	29.7	19.9	1
1	26.3	19.0	19.2	15.7	9.1	15.4	22.6	21.6	42.1	47.7	2
1	53.6	41.9	51.9	53.9	42.6	36.6	37.8	36.5	28.8	43.3	3
1	37.8	28.5	25.4	32.6	21.7	22.4	27.5	41.4	42.5	39.1	4
1	48.3	45.2	45.5	50.8	44.4	43.4	41.1	41.8	36.4	34.2	5
1	42.0	31.9	30.0	26.1	32.9	26.3	25.5	38.0	39.7	29.9	6

The first number (2) indicates the source of data, you may leave it unchanged. The second number (1) indicates the unit system, which should be in accordance with the **Output Units** setting of the Project. In this case, units are customary (1 - customary, 2 - metric). The third line contains location. The fourth line contains precipitation monthly totals and temperature averages and the remaining lines contain daily values. The first number in a row is a relative year #, the next ten are daily values, the last number in a row is a number of the row within a year (37 rows for a year).

A characteristic part of the Solar Radiation file is presented below.

SOLAR RADIATION

2

1

Cheyenne WY

41.15

1	212.9	221.2	250.5	128.6	183.6	178.2	226.3	202.7	242.2	170.4	1
1	175.6	188.0	86.0	239.2	229.9	170.7	118.1	55.7	181.1	268.6	2
1	252.7	189.0	229.9	171.0	276.8	273.3	303.8	307.0	162.6	188.6	3
1	148.7	227.3	260.5	246.0	197.4	243.6	310.8	342.6	270.0	193.0	4
1	288.5	358.6	347.4	285.6	364.6	336.4	224.9	271.8	159.8	343.6	5

The only difference here is that in row four there is a latitude of the site instead of precipitation monthly totals or temperature averages.

In addition, the following is a file containing evapotranspiration parameters:

EVAPOTRANSPIRATION

1

Cheyenne WY

41.15 138 273 2.5 12 12.9 52 54 50 51

The first two lines provide information about the unit system and location. The third line contains numbers for:

Site latitude, growing season start day, growing season end day, maximum leaf area index, maximum evaporative zone depth (in), average annual wind speed (mi/hour), and average relative humidity for the four quarters of a year.

This is the way you should format your data. Note that in HELP you may use only daily data for full years.

To be able to run the model using the new data:

1) Run the Weather Generator with parameters of one of the database weather stations. Set the number of years for simulation equal to the number of full years of real data you have converted into HELP format.

2) Import your data into the project directory with the names (replace existing files for the dummy weather station):

_weather1.dat

_weather2.dat

_weather3.dat

_weather4.dat

3) Run the HELP Model again.

You can import and effectively use your own weather data. However, this option is not very effective if you do not have a full set of data. Say, you have only precipitation and temperature but don't have solar radiation. For the latter case you should use the precipitation and temperature correction option of Weather Generator.

Note: You cannot utilize both, the PRECIPITATION/TEMPERATURE tab and the EVAPOTRANSPIRATION tab, for this purpose (corrections made with these tabs work for a single run only).

To use the precipitation, temperature, or evapotranspiration correction option:

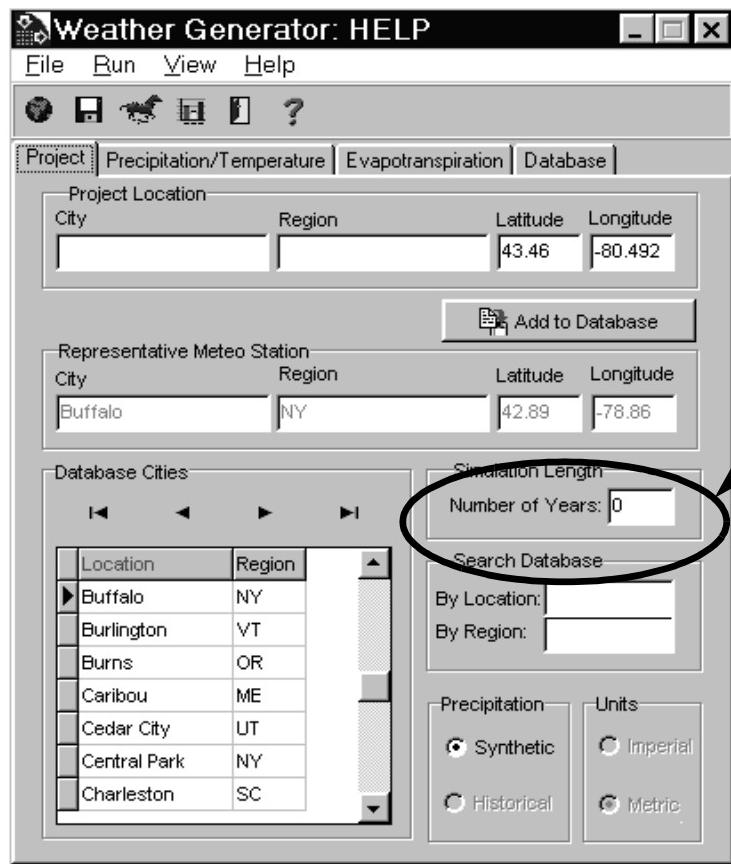
- 1) Choose a weather station from the database nearest your conditions and add it to the database with a new name (type your project location and region under the **Project Location** of the **Project** tab of **Weather Generator** and click **Add to Database** button. The **Database** tab will activate and the asterisk will show the newly added record with the name which you have input).
- 2) Scroll the bottom bar until the **Default normal mean monthly temperature T1(Jan)[F]** column appears in the dialogue box. Input the average value for January (degrees Fahrenheit) for your location, scroll the bar further and edit the rest of the mean monthly values for temperature.
- 3) Just after mean monthly temperatures you will find the **Default normal mean monthly rainfall R1(Jan)[IN]** column which contains mean monthly total precipitation for January. Input the average value for January (in) for your location, scroll the bar further and edit the rest of the mean monthly precipitation totals.
- 4) Scroll the bottom bar to the left and find **Growing season start day**, **Growing season end day**, **Default maximum leaf area index**, and three values of **Evaporative zone depth** for different types of vegetation cover. Edit these values if required.
- 5) Scroll the bottom bar to the right and find **Mean annual wind speed [mph]**. Edit these values if required.
- 6) Scroll the bottom bar to the far right and find four values of the **Quarterly mean relative humidity (RHUM)** for the four quarters of a year. Edit these values if required.
- 7) Click the **Save** icon (checkmark symbol) above.
- 8) Now you can use this database record as a project weather station (customized to your local conditions) with the whole set of features associated with the **Weather Generator**.

7

Running the Model, Viewing Output, and Reporting

Setting the Simulation Time with the Weather Generator

The simulation time is limited by the amount of weather data prepared. The model will run for a length of time that weather data is available. Likewise, when you use the Weather Generator to create statistically reliable data, you must type the simulation time in the **Number of Years** box, which is located in the **Simulation Length** frame on the Weather Generator dialogue box shown below:



Simulation Length text box - enter simulation length in years.

The default simulation time is zero years. You must type in a positive integer value to use the Weather Generator. The maximum number of simulation years is 100. The minimum number of years is one.

To run the Weather Generator and set the simulation time:

From the **Run** menu  **Weather Generator**, or 

The **Weather Generator** dialogue box will appear.

Type the simulation time in the **Number of Years** box, which is found in the **Simulation Length** frame.

Specify the other parameters for your landfill site in the Weather Generator dialog box.

 **Run** to generate weather data, or 

Click **File**, and  **Save** to save your data, or 

Click **File**, and  **Exit** to close the Weather Generator, or 

Statistically reliable weather data has been generated, and the simulation length has been set.

Running the Visual HELP Model



To run the model for a single profile click the profile icon above the Profile View.

To run the model for multiple profiles or for one profile if it is a single project:



- Click the same icon from the Operational toolbar above the Project Tree View

or

-  **Run** (in the main menu). 

The program will collect input files and run the model. A box with a progress bar will appear.

Interpreting the Output and Preparing a Report

Original DOS HELP Output

UnSat Suite allows you to view and print the original DOS HELP report. If you are familiar with original HELP output, you may choose to use these results.

To view and print original HELP output:

☞ **Output** from the main menu.

☞ Original Listing.

The **Original Model Listing** dialogue box will appear. Here you can view the original listing, find specific expressions, mark and print parts or make a printout of the model results.

If you want to see the HELP input file, click the **Input File** tab.

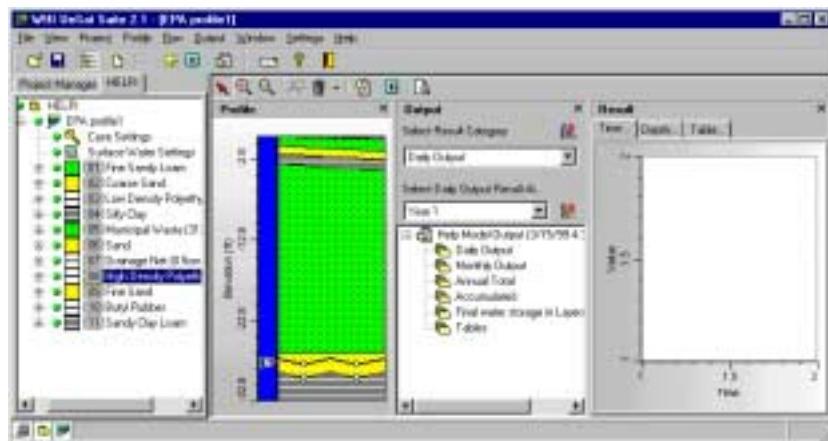
To print the file or the marked part of a file:

☞ **Print** from the **File** menu.

Specify the print properties, and click [OK] to print.

Viewing the Output Graphs

After the model has successfully ran, the Output View and Result View windows will open and the UnSat Suite window will look the following way:

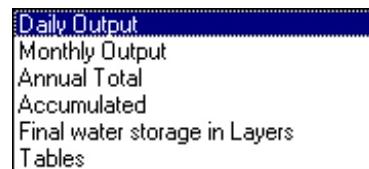


To enlarge the graphs viewing area you may:



Click the icon to close the Project Tree View, or click the 'X' in the Profile View to close it.

To select the output category, click the arrow in the **Select Result Category** drop-down listbox. The following list will appear:



Click the category you wish to view.

The first possible result group will appear in the listbox below. To view all available result groups, click the arrow in the **Select 'Name of Category' Result at...** drop-down listbox.

Annual Total and Accumulated Annual Balance

UnSat Suite offers a wide range of methods of viewing results. We recommend that you begin by viewing your results with an **Annual Total** time scale.

The option to graph **Accumulated Volumes** is a new feature, offered only in UnSat Suite. This plot will show you the total volumes of water that flowed or were drained during a period of time. This allows you to examine the total volume of leachate to percolate through the landfill bottom during a specified period of time and assess total volumes of other water balance constituents.

The list of available types of balance will appear in the lower listbox if you have selected **Annual Total** or **Accumulated** in the upper **Select Result Category** drop-down listbox: **rate**, **volume**, and **percent (Annual Total)**.

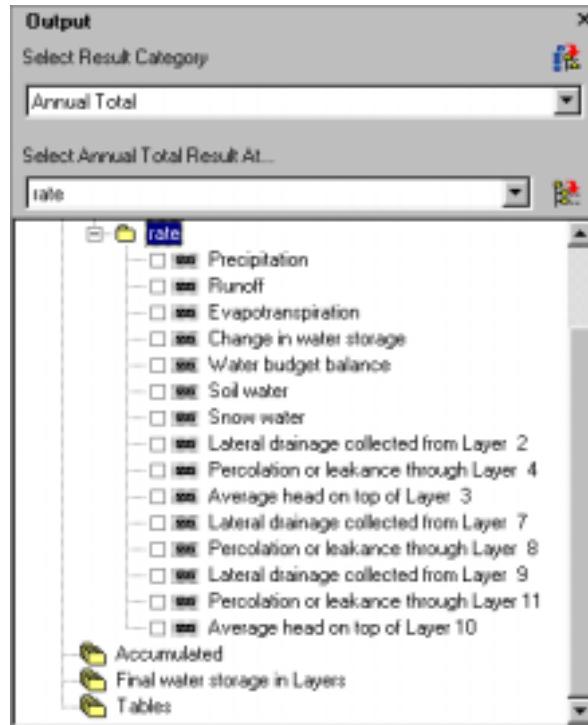
Here **rate** means annual rate of the balance constituent, **volume** means volume for the area represented by the profile and **percent** means percent of the ongoing volume of water which is precipitation.

Select **rate**.

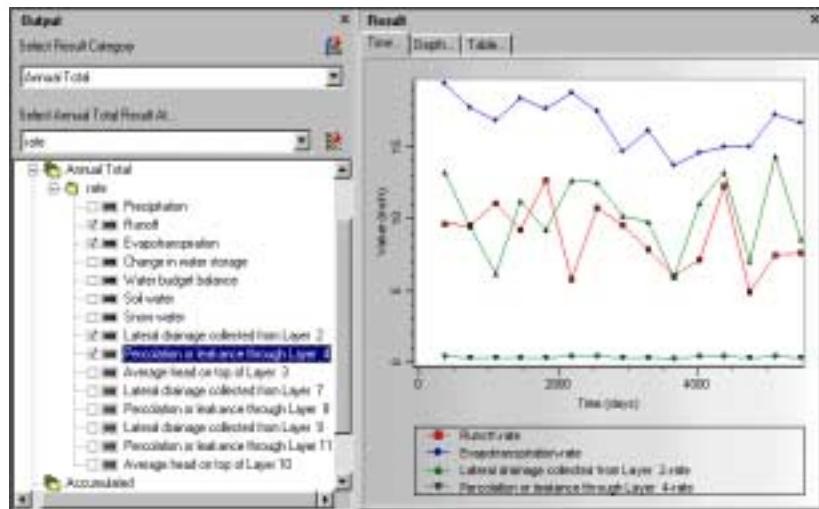


To view all results available for this specific type of balance, click the icon to the right of the **Select Annual Total Result at...** box.

The list of available balance constituents will open in the Result Tree:



Click the check boxes beside the types of variable you wish to view. The graph of the variable will appear in the Result View window:



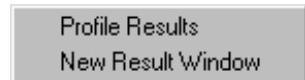
To erase a variable from the Result View window, deselect the corresponding check box in the Result Tree.

If you wish to clear the Result View window:

☞ **Output** from the main menu.

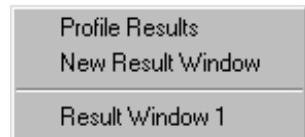
☞ **Clear Display Results**.

To view many variables, you may place results for some variables into a new Result Window. <right click> the name of the variable you wish to put into a new window. The following menu will appear:



Choose **New Result Window**. Results for the variable will appear in the **Result Window 1**. You may add graphs for the other variables to the **Result Window 1** using the same method.

To see outputs for many variables, open additional Result Windows and place results there. <right click> the name of the additional variable. The following menu will appear:

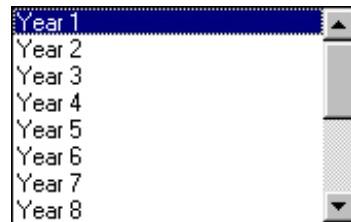


1. Choose New Result Window. Results for the additional variable will appear in the **Result Window 2**. You may repeat these steps.

All viewing methods described above may be applied to study **Accumulated** annual water balance constituents.

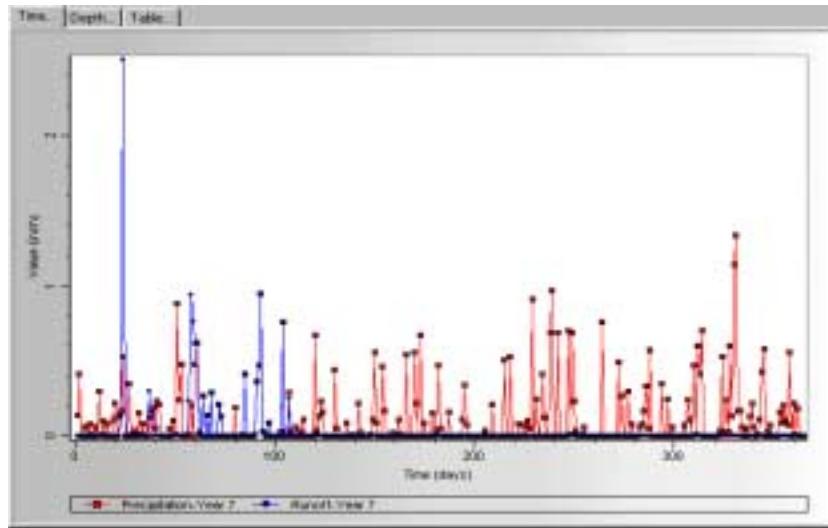
Daily and Monthly Output

If, from the **Select Result Category** you select **Daily Output** or **Monthly Output**, you will be prompted to select the year for which you would like to have results to be displayed from the lower listbox:



 Scroll the list of years, if necessary, and select the desired year. To view all results available for this specific type of balance, click the **Add to Output Tree** icon to the right of the lower output listbox.

Select the desired types of balance variables the same way it was done for the Annual Total category and view results:



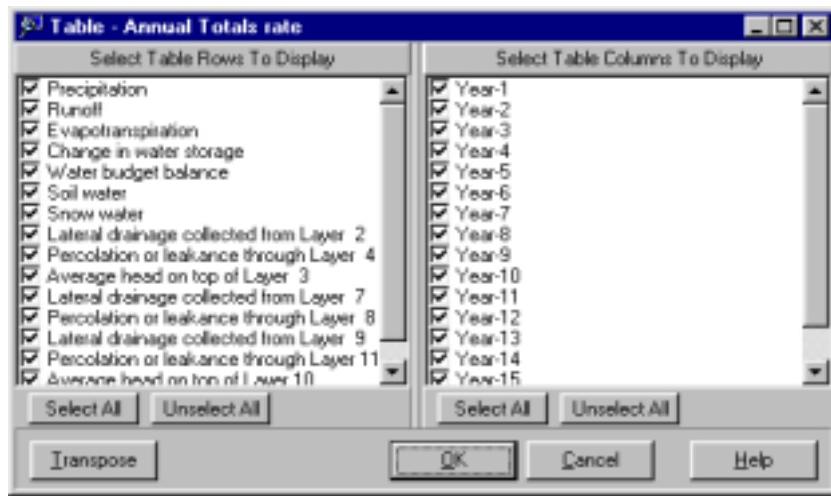
All viewing methods described above for **Annual Total** balance constituents may be applied to study **Daily** and **Monthly** output categories.

Viewing HELP Tables

In addition to UnSat Suite's useful graphing capabilities, you can also obtain information from **Annual Total**, **Accumulated** balance and **Peak daily values** of different water balance constituents in table format.

To view HELP tables:

- ☞ **Tables** from the **Select Result Category** drop-down listbox.
 - ☞ the type of table you wish to view from the lower listbox and click the **Add to Output Tree** icon to the right of the lower output listbox.
- The table will appear in the Output Tree. Click the check box to the left of the table's name to open the list of available rows and columns.



Here you may select desired output times and variables to customize your table. You may use the following tools for editing a table:

- ☞ **Unselect All** to unselect all times or variables and then click desired if you want to show only a small number of rows or columns in the table.
- ☞ **Select All** if you wish to specify all lists after you have unselected some times or variables.
- ☞ **Transpose** if you want to switch columns and rows.
- ☞ **OK** after you have set a table. The table will appear in the Result View:

	Year1 [inch]	Year5 [inch]	Year10 [inch]	Year15 [inch]
Precipitation	4.2980E+01	3.8300E+01	3.3700E+01	3.9770E+01
Runoff	9.5340E+00	9.5540E+00	1.1088E+01	9.2324E+00
Evapotranspiration	1.9387E+01	1.7692E+01	1.6730E+01	1.8309E+01
Lateral drainage collected	1.3191E+01	9.4630E+00	6.1681E+00	1.1183E+01
Percolation or leakage	4.0717E-01	3.5288E-01	2.9935E-01	3.9890E-01

All methods of table editing are applicable to the **Peak daily values** table: This table differs in its components from the rest of the tables.

However, the data it contains allows you to focus at the extreme hydrologic events.

Peak daily values				
	Rate (inch)	Volume (ft ³)	Day	Year
Precipitation	2.4500E+00	8.8933E+03	234	12
Runoff	2.7828E+00	1.0101E+04	76	2
Lateral drainage collected	5.9612E-01	2.1639E+03	295	14
Percolation or leakage	8.0728E-03	2.9304E+01	295	14
Lateral drainage collected	7.7800E-03	2.8241E+01	295	14
Percolation or leakage	4.4826E-08	1.6271E-04	295	14
Lateral drainage collected	4.2637E-08	1.5439E-04	332	7
Percolation or leakage	2.3721E-08	8.6106E-05	218	14
Snow water	7.5636E+00	2.7477E+04	68	3

Note that the Rate indicated is a unit rate based on the layer (i.e. in the above example, 2.45 inches of precipitation fell everywhere in the model). To determine the volumetric rate, multiply this value by the area you are interested in.

Explanation of Variables

The variables that can be graphed in Visual HELP are explained below:

Precipitation	Inflow in the form of rain or snow.
Runoff	Water from precipitation that does not infiltrate into the landfill, and flows off at the surface.
Evapotranspiration	Evaporation from the leaves and soil surface, as well as transpiration by plants.
Evaporative zone water	Water storage that can be extracted by evapotranspiration. The evaporative zone depth is specified in the Weather Generator.
Change in water storage	Total change in the amount of water stored in the profile.
Annual water budget balance	Inflow water minus outflow water minus the change in water storage.
Soil water	The amount of soil water at the end of the year.
Snow water	The amount of snow water at the end of the year.
Lateral drainage from layer ‘x’	The amount of water drained, by pipe or slope drainage, from the lateral drainage layer ‘x’.

Percolation or leakage through layer ‘x’ The amount of water percolated through the barrier soil liner ‘x’, or through a geomembrane liner if it is not underlaid by a barrier soil liner.

Average head on top of layer ‘x’ The mean head on top of a liner.

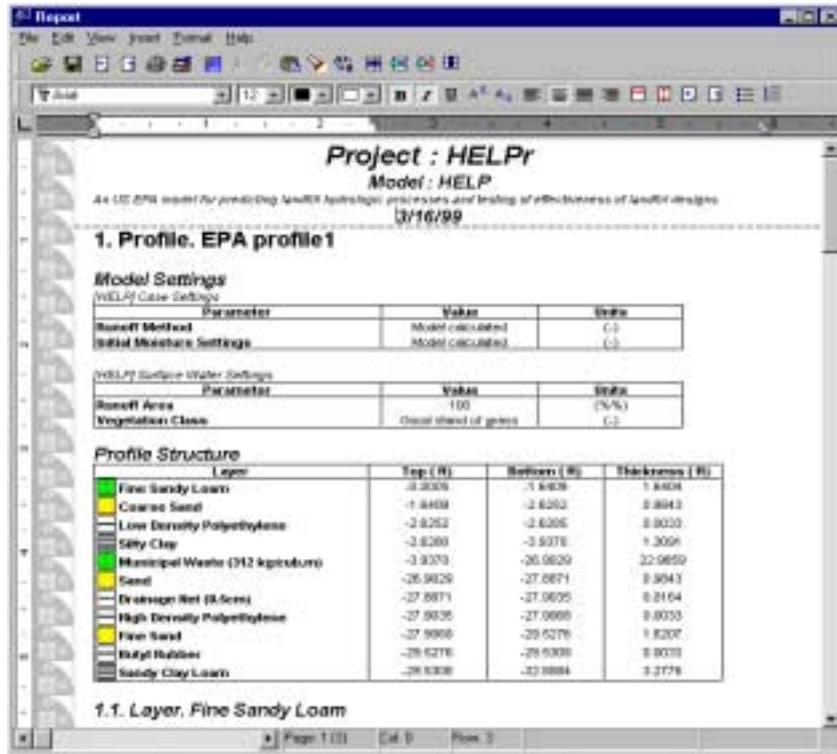
Deviation of head on top of layer ‘x’ The standard deviation of the head on top of a liner.

Note: For Daily and Monthly time scales, the only available output units are millimeters (metric system) and inches (customary system) or other length units which you may set with your Units template. These units measure the water balance constituent as a head per unit area of the landfill. For the Annual Total and Accumulated Volumes, you can also view the results in cubic meters or cubic feet. These units measure the water balance constituent as a volume of water in the part of the landfill represented by profile. For the Annual Total, you can also plot the results as a percentage of the total inflow of water.

Creating a Report

To present results of your Visual HELP simulation to your clients, use the UnSat Suite Report Generator.

 To create a report and add the project input data to it, click the icon from the Operational Icons toolbar. The report will appear in a separate window:



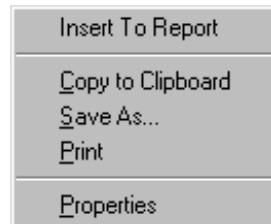
By default, the Report Generator lists all input data for your project.

In the **Report** window you may edit the report, input your own text and add any type of graphics or table outputs produced by UnSat Suite.

Note: The graphs and tables will be placed at insertion point.

To add a graph or table to the report:

- [1] In the **Report** window place the cursor to the position you want your graph or table to appear in the report.
- [2] Create a graph or table using one of the methods described above.
- [3] <right click> in the Result View. The following menu will appear:



- [4]  **Insert To Report.** The graph or table will appear in the report.

A graph may appear a size smaller than the original. To view the graph of desired size, click the graph in the **Report** window and stretch it until it reaches proper size.

A table may be longer than the Report window allows. In this case the table will be automatically wrapped.

Add necessary graphs and tables in the report and write your comments. You may insert a header and footer into your report. Apply different fonts and styles while working in the **Report** window. To utilize these and other options, make corresponding selections from the top menu. After you are done, you may print the report or save it.

Part 3:

The PESTAN Model

Introduction

PESTAN is a popular US EPA model for evaluating the environmental impacts of potential non-point agricultural sources of groundwater contamination. It simulates the 1-D vertical transport of organic pollutants, commonly pesticides, through homogeneous soil to groundwater. The model is based on a close-form analytical solution of the advection-dispersive-reactive transport equation.

The model was developed for initial screening assessments to evaluate the potential for groundwater contamination of already registered pesticides and those submitted for registration. The model has been tested under field and laboratory conditions (see extracts from the original PESTAN manual in Appendices).

Allowed profile structure: One-layer homogeneous profile.

Simulated processes:

Surface: Constant recharge rate, agricultural application of pesticides.

Subsurface: Flow and transport of pesticides through the soil with constant velocity, sorption and decay of pesticide, leaking of pesticide to groundwater.

The vertical transport of dissolved pollutant through the vadose zone is simulated in PESTAN as a 'slug' of contaminated water that migrates in homogeneous partly saturated soil. The concentration of the chemical slug equals to the solubility of the pollutant in water.

A maximum of ten applications of the active ingredients can be applied in a single calculation and for each application, the time of application prior to recharge needs to be provided.

The slug begins to enter the soil at the first precipitation and irrigation event at a rate equal to the pore water velocity. The pollutants stored at the soil surface before the recharge is subject for solid-phase decay. Once the recharge starts, the remaining pollutant is considered dissolved and starts to enter the soil. In the soil the pollutant is influenced by liquid-phase decay, sorption and dispersion. The flow of the pollutant slug occurs with the constant velocity. The hydraulic conductivity of the soil accounts for partly saturated conditions using the Campbell's equation.

These are the basics of PESTAN conceptualization and assumptions. For more information on the model's theory, see extracts from the original PESTAN manual in Appendices.

Assumptions incorporated in PESTAN do not allow it to simulate such sophisticated cases as VS2DT allows. However, this model proved to be a very efficient tool for preliminary estimates of environmental impacts of potential non-point agricultural sources of groundwater contamination. The potential of this model is satisfactory in a wide range of

environmental expertise and it has a lot of advantages to be used as a base model for risk assessment studies which require multivariant simulations.

UnSat Suite allows you to quickly prepare, run and interpret PESTAN simulation by the use of graphical tools. PESTAN is also equipped with a limited database of soil materials and pesticides. It is planned that in the next release of the UnSat Suite the database for several hundred of existing pesticides will become available.

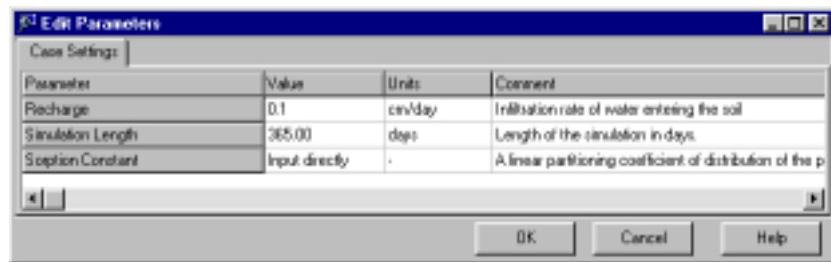
8

Input Specification

The Case Settings

The following parameters are editable under the **Case Settings** group in PESTAN:

- Recharge rate
- Simulation length
- Sorption Constant



To edit the case settings:

- 1) the **Case Settings**,
- 2) Or <right click> the **Case Settings**.
- 3) Click **Edit**.
- 4) Type in the new value for the appropriate characteristic in its **Value** box.
- 5) Adjust the units in the **Units** list.
- 6) Click **[OK]**.

The following parameters can be edited using the methods discussed above:

Recharge

Infiltration rate of water entering the soil.

Simulation Length

The total time length of simulation.

Editing Sorption Constant

Sorption Constant is a qualitative parameter. The selection made for this parameter affects data requirements of other parameter groups.

The sorption constant is the linear partition coefficient which describes the relative distribution of the pollutant between the solid phase and the dissolved phase in water. It is a function of the fraction of organic content of the soil and can be estimated as the product of the fraction organic content (characteristic of soil) and the organic carbon partition coefficient of the pollutant (characteristic of pollutant). The fraction organic content and the organic carbon partition coefficient of the pollutant are much more accessible parameters than the sorption constant. However, PESTAN requires just the value of the sorption constant. The UnSat Suite allows the user to input this value either directly or to input values for its more available components.

To select the way of inputting the sorption constant, in **Case Settings** click in the **Value** column beside it.

Click either **Calculate** or **Input Directly**.

Click **[OK]**.

The following chart shows how the choice made in the Case Settings can affect the other parameters:

Selection	Location of New Parameter	New Parameter
Calculate	Soil Parameters	Fraction Organic Content = Fraction of organic content in the soil.
Calculate	Pesticide Parameters	Organic Carbon Partition Coefficient = Chemical characteristic of the chemical.
Input Directly	Soil Parameters	Sorption Constant = The linear partition coefficient describing distribution of the pollutant between the solid phase of soil and pore solution.

The new parameters are all numeric values and can be edited directly at their respective locations.

Time Dependent Groups

There are two time dependent groups of parameters in PESTAN:

Waste Application Schedule Times and rates of chemical applications prior to recharge start. The pesticide is a subject to the solid phase decay during the storage time.

Observation Times The desired times for output.

To open the time dependent parameter groups:

- [1] the group name,
- [2] Or <right click> the group name.
- [3] Click **Edit**.

Waste Application Schedule

To add times of pesticide application [Add].

Alter the values for **Time** and **Waste Application Rate** in the appropriate boxes. Click **Add**, the value of **Waste Application Rate** will be copied and the value of **Time** will grow by the value of the previous time step (the difference between two previous application times). If you wish to input pesticide application schedule with variable time steps and rates, edit appropriate cells. To edit the pesticide application schedule you may use the following tools:

To insert a time step between two existing time steps:

- 1) Click the boxes above where your new step will be inserted.
- 2) [**Insert**].
- 3) Change the time and rate.

As easy as time steps can be added, they can also be deleted.

To delete a time step:

- 1) Click the box for the step you wish to delete.
- 2) [**Delete**].

All the time steps can be deleted by [**Delete All**].

[**OK**] after you are done with the input.

An example of a pesticide application schedule is shown below:

Edit Parameters

Waste Application Schedule

Parameters	Time	Waste Application Rate
Units	days	kg/ha
Comments	Enter your time values in this column	Rate of chemical application. Time is counted prior to start of simulation.
1	0	10
2	30	10
3	60	20

Add **Insert** **Delete** **Delete All**

OK **Cancel** **Help**

Observation Times

All methods of inputting and editing of the **Waste Application Schedule** are applicable for **Observation Times**. The default value in the **Is Output Desired?** field is **yes**. If you do not want output to be recorded for specific time this run, click in the appropriate cell and deselect the check box. The cell value will turn to **no**.

Edit Parameters

Observation Times

Parameters	Time	Is Output Desired?
Units	days	-
Comments	Enter your time values in this column	Select 'yes' if you want output to be documented at this time. Otherwise check 'no' in appropriate line.
1	0	yes
2	10	yes
3	20	yes
4	30	yes
5	60	no
6	90	yes
7	120	yes
8	150	yes

Add **Insert** **Delete** **Delete All**

OK **Cancel** **Help**

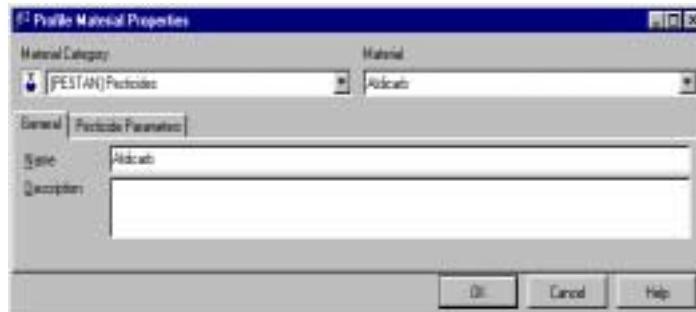
Specifying the Pesticides

Substituting the Pesticide

To substitute a chemical, open **Profile Material Properties** dialogue box.

To open Profile Material Properties dialogue box:

- 1) the **name** of the pesticide
- Or
- 2) the **name** of the pesticide and click **Edit**.



the drop-down arrow of the **Material** listbox. The list of available pesticides will appear:



Select a desired pesticide and click it. After you have selected the pesticide, you may edit its parameters to make them completely match your case.

In the UnSat Suite version 2.2.02 the list contains only 15 sample pesticides. It is planned to add several hundred common pesticides to the database for the next version of UnSat Suite.

Editing Pesticide Properties

You can access the properties of a pesticide in two ways:

To access pesticide properties:

- 1) the **name** of the pesticide
 Pesticide Parameters tab
- Or
- 2) the **name** of the pesticide and click **Edit**
 Pesticide Parameters tab

General Pesticide Parameters			
Parameter	Value	Units	Comment
Water Solubility	99.00000	mg/l	Solubility of the chemical in water under the standard conditions.
Organic Carbon Partition Coefficient	999.000000	ml/g	Coefficient describing the partitioning of the chemical between organic and aqueous phases.
Solid Phase Degradation Rate Const.	0.000100	/hr	A parameter of chemical decay in solid phase.
Liquid Phase Degradation Rate Const.	0.000100	/hr	A parameter of chemical decay in liquid phase.

The following is a list of changeable pesticide characteristics and a brief description of each:

Water solubility

The solubility of the chemical in water under the standard conditions.

Organic carbon partition coefficient Characteristic organic carbon content of the soil.

Solid-phase degradation rate constant Decay of the pollutant at the surface occurs prior to the infiltration into the soil starts. Decay is defined as the rate of loss per hour.

Liquid-phase degradation rate constant Liquid-phase decay describes the process where mass is lost within the soil system. Degradation occurs primarily by soil microorganisms and may vary depending upon soil temperature and moisture.

Modifying the Profile

Profile Properties

To view or edit the profile properties:

<right click> the profile picture, and click **Profile Properties**

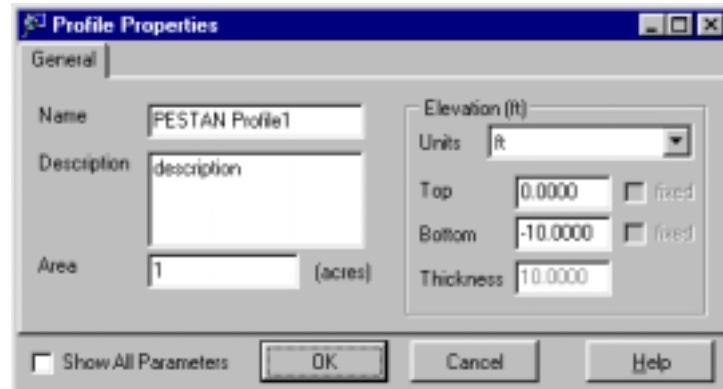
Or

<right click> the profile name

Or

<right click> the profile name and click **Profile Properties**

The **Profile Properties** dialogue box will appear:



The following data are available in the **Profile Properties** dialogue box:

Name Name of the profile.

Description Description of the profile.

Elevation You may specify the elevation of the profile top or bottom in this part of the dialogue box.

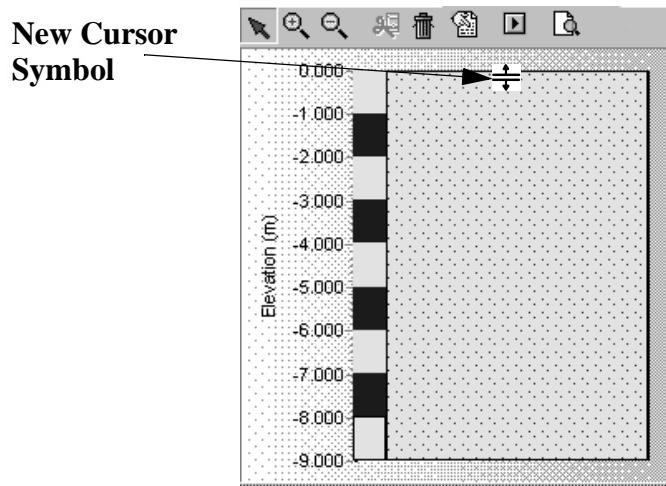
Area Type the land area represented by this profile in this box.

Resizing the Layer

You may resize the layer either through the **Profile Properties** dialogue box by changing its top and bottom elevation or graphically in the Profile View.

To resize a layer graphically:

- 1) Move the mouse pointer to the layer's boundary. The pointer symbol will change.

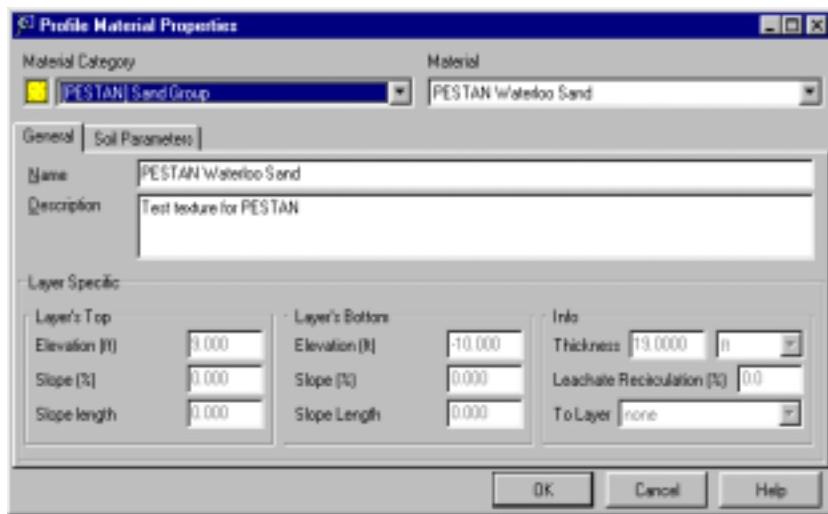


- 2) Click and drag the boundary to its new location.
- 3) Type the correct elevation in the **Confirm Value** dialogue box.
- 4) Click **[OK]**.

Substituting the Layer

To access a layer:

- 1) <right click> on the soil in the Profile View.
- 2) Click **Layer/Properties**. The **Profile Material Properties** dialogue box for the soil will open,
or
☞ the layer name in the Project Tree View,
or
<right click> the layer name and click **Properties**.



To substitute a layer:

- 1) Select a new material category from the **Material Category** list. The available soils will appear in the **Material** drop-down listbox.
- 2) Select a new soil (material) from the **Material** list.
- 3) Give the new layer a unique name and write a descriptive comment.
- 4) Click the **Soil Parameters** tab and edit the values for soil parameters if necessary.
- 5) Click **[OK]**.

Editing Soil Properties

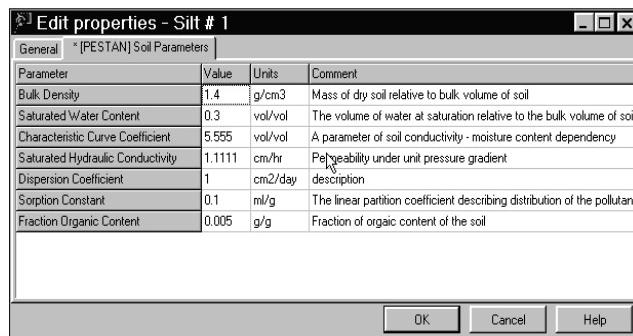
Open the **Profile Material Properties** dialogue box using one of the methods described in the previous section.

☞ the **Soil Parameters** tab.

Parameter	Value	Units	Comment
Bulk Density	1.4	g/cm ³	Mass of dry soil relative to bulk volume of soil
Saturated Water Content	0.3	vol/vol	The volume of water at saturation relative to the bulk volume of soil
Characteristic Curve Coefficient	5	-	A parameter of soil conductivity - moisture content dependency
Saturated Hydraulic Conductivity	1	cm/hr	Permeability under unit pressure gradient
Dispersion Coefficient	1	cm ² /hr	Characteristic of the ability of soil to disperse the flow of chemicals
Sorption Constant	0.1	ml/g	The linear partition coefficient describing distribution of the pollutant between soil and water
Fraction Organic Content	0.5	%	Fraction organic content of the soil in percent

Under the **Soil Parameters** tab, you can edit the properties of the layer material.

To edit a value of a property, select units (if necessary), click in the **Value** box and type your changes. The changes will be saved when you click **[OK]**.



The following is a list of changeable soil characteristics and a brief description of each:

Bulk Density	The mass of dry soil relative to the bulk volume of soil.
Saturated Water Content	The saturated water content of the soil is the volume of water at saturation relative to the volume of soil. This parameter is equal or slightly less (because of entrapped air), than the soil's porosity.
Characteristic Curve Coefficient	The parameter of the dependency of the relative conductivity of the soil to the relative saturation under steady-state conditions in Campbell's equation. May vary from 4 for sand to 12 for clay.
Saturated Hydraulic Conductivity	Permeability of saturated soil under a unit pressure gradient.
Dispersion Coefficient	Coefficient of hydrodynamic dispersion.
Sorption Constant	Describes the ratio of the pollutant that is sorbed to the solid phase to the portion that is dissolved in water. The higher the value of the partition coefficient the greater the sorption to the solid phase.
Fraction Organic Content	Fraction of organic content in the soil expressed as a percent.

Note: Depending on your selections in the Case Settings group, either

Sorption Constant or Fraction Organic Content will appear in Soil Parameters group.

For more explanation on soil parameters, see PESTAN original manual extract in the appendices.

Observation Point

UnSat Suite allows you to view output at 100 different depths with a constant depth step within the profile. However, you may want to view results at a specific depth which may not coincide with one of the default depths. This need particularly increases when you are using data obtained from a probe set at specific depth for site model calibration. For this purpose you may set an observation point to measure the pesticide concentration at any depth in the profile. Only one observation point is allowed in PESTAN.

Adding Observation Points

Please NOTE: At present, this feature is not enabled in PESTAN. The information contained below has been included as a reference for future releases of UnSat Suite.

To select the position of observation points:

<right click> on the Observation Points Column to the right of the profile picture in the Profile View.

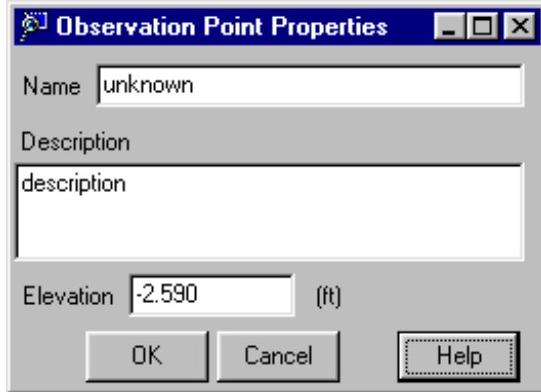
☞ **Add**

Or

<right click> on the Profile Name in the Project Tree View.

☞ **Add Observation Point**

The following dialogue box will appear:



Enter the name in the **Name** text box and the depth, where you wish to place your observation point, in the **Elevation** box.

☞ [OK]

The observation point will appear in the Profile View and in the Project Tree View.

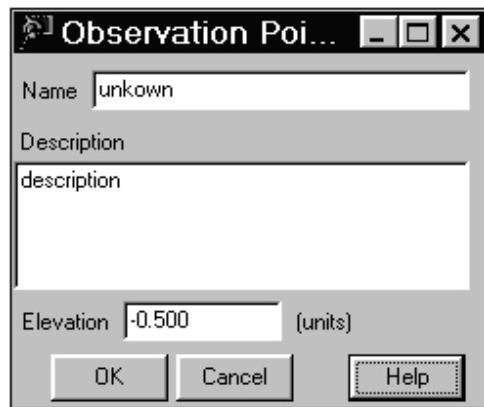
Observation Point Properties

To change the properties of an observation point:

<double click> on the observation point in the Profile View.

Or <right click> on the observation point name in the Project Tree View and select **Properties**.

The following dialogue box will appear:

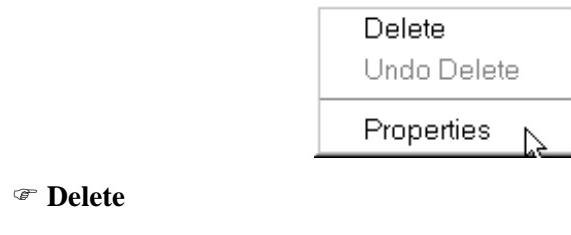


Edit properties of the observation point and ☞ [OK].

Deleting an Observation Point

To delete an observation point:

<right click> on the observation point in the Project Tree View.



☞ Delete

Restoring an Observation Point

To restore an observation point:

<right click> on the observation point in the Project Tree View.

☞ Undo Delete

The observation point will appear in the Profile View and in the Project Tree.

9

Viewing Output and Reporting

Original DOS PESTAN Output

UnSat Suite allows you to view and print the original DOS PESTAN output. If you are familiar with original PESTAN output, you may choose to use these results.

To view and print original output:

- ☞ **Output** from the main menu.
- ☞ **Original Listing.**

The **Original Model Listing** dialogue box will appear. Here you can view the original listing, find specific expressions, mark and print sections or make a full printout of the model results.

If you want to see the PESTAN input file, click the **Input File** tab.

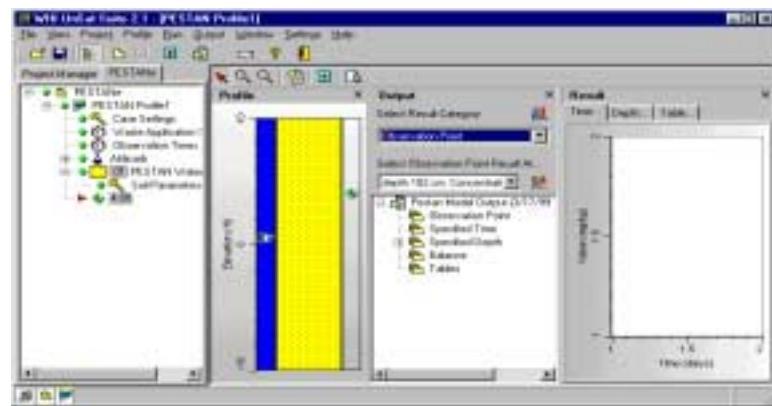
To print the file or the selected part of a file:

- ☞ **Print** from the **File** menu.

Specify the print properties, and click **[OK]** to print.

Viewing the Output Graphs

After the model has successfully ran, the Output View and Result View windows will open and the UnSat Suite window will look the following way:

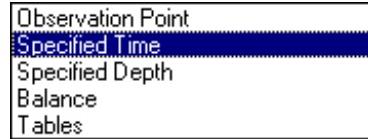


To enlarge the graph viewing area you may:



Click the icon to close the Project Tree View, or click the 'X' in the Profile View to close it.

To select the output category, click the arrow in the **Select Result Category** drop-down listbox. The following list will appear:



Click the category to view.

The first available result group for this category will appear in the listbox below. To view all available result groups, click the arrow in the **Select 'Name of Category' Result at...** drop-down listbox.

Specified Time and Depth

The list of all times, including the observation times, will appear if you selected **Specified Time** in the **Select Result Category** drop-down listbox:

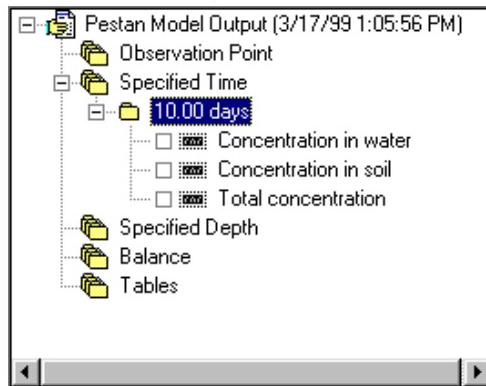


Use the slider to reach the time of your interest and click it. The selected time will appear in the drop-down box:



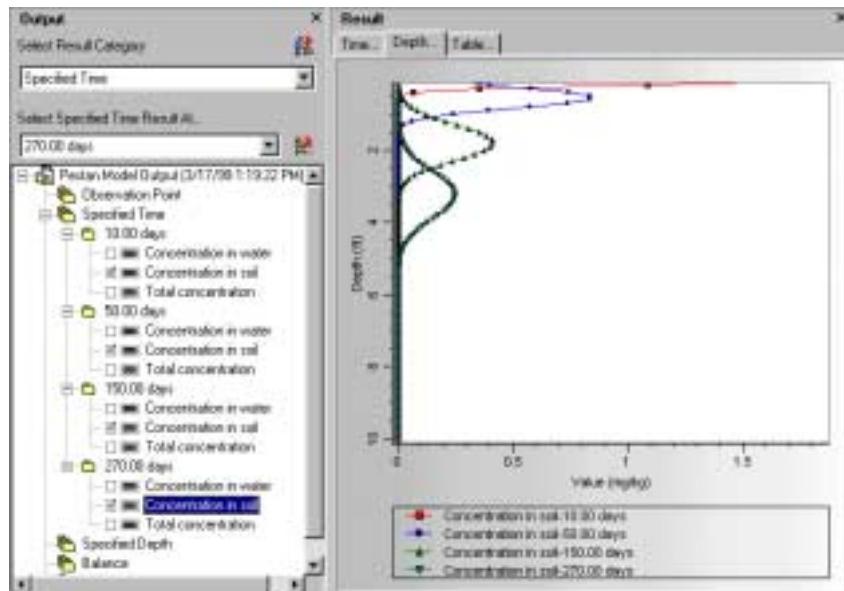
To view all results available for this specific time, click the icon to the right of the **Select Specified Time Result at...** box.

The list of available results will open in the Result Tree:



Click the check box beside the type of variable you wish to view. The graph of the variable will appear in the Result View window.

To add the graph for another times to the same window, select a new times from the **Select Specified Time Result at...** box and check the same variables (you will get a warning if you choose different variables). The Results View will show profile distribution of the variable for different times:

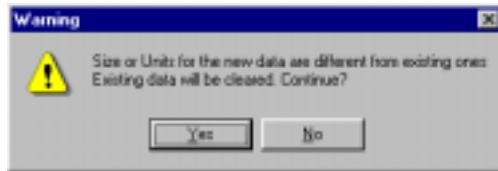


To erase an output for specific time from the Result View window, deselect the corresponding check box in the Result Tree.

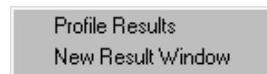
If you wish to clear the Result View window:

- ☞ **Output** from the main menu.
- ☞ **Clear Display Results.**

To view output for other variables, click the corresponding check box. A warning will be posted if the new and previous variables are measured in different units:

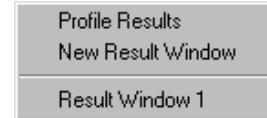


To view both variables, which are measured in different units, you may place results for the second variable into the new Result Window. To do this, <right click> the name of the second variable. The following menu will appear:



Choose **New Result Window**. Results for the second variable will appear in the **Result Window 1**. You may add graphs for the other times to the **Result Window 1** using the same method.

To see outputs for more than two variables, you may open additional Result Windows and place results there. To do this, <right click> the name of the additional variable. The following menu will appear:



Choose **New Result Window**. Results for the second variable will appear in the **Result Window 2**.

All the methods described for the **Specified Time** output viewing, are applicable to the **Specified Depth** category. The graphs of this category will present time changes of the variable occurred at a specific depth.

Points of the profile depth are restricted to 100 nodes. The list of all nodes will appear if you selected **Specified Depth** category in the **Select Result Category** drop-down listbox and click the drop-down arrow of the **Select Specified Depth Result at...** box:



Select desired depth from this list, open the list of available variables and select variables to view using the same tools which were described earlier for the **Specified Time** output category.

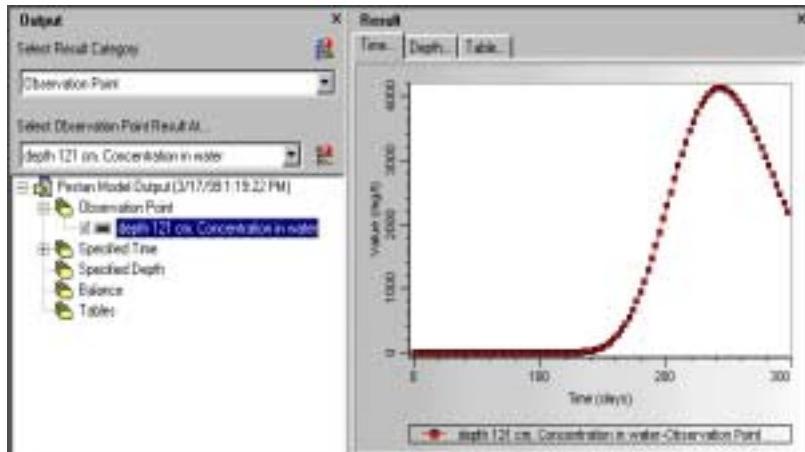
Observation Point

To view the graph of the pesticide concentration at observation point, select **Observation Point** from the **Select Result Category** drop-down listbox.



Click the icon to the right of the **Select Specified Time Result at...** box to add the output to the Output Tree.

Select the check box beside the variables name and view the results:



Balance

The PESTAN model allows you to compute the balance of the profile. In UnSat suite you may view the balance graphs and add them to a report.

Select Balance in the **Select Result Category** drop-down listbox.

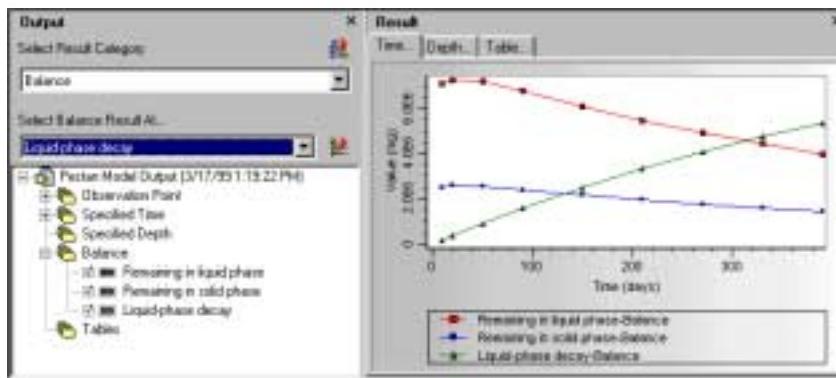


To view all results available for this specific type of output, click the icon to the right of the **Select Balance Result at...** box.

The list of available variables will appear:

- Remaining in liquid phase
- Remaining in solid phase
- Total mass of remaining
- Liquid-phase decay

Select the desired variables and view them by using all tools described earlier.



Viewing Tables

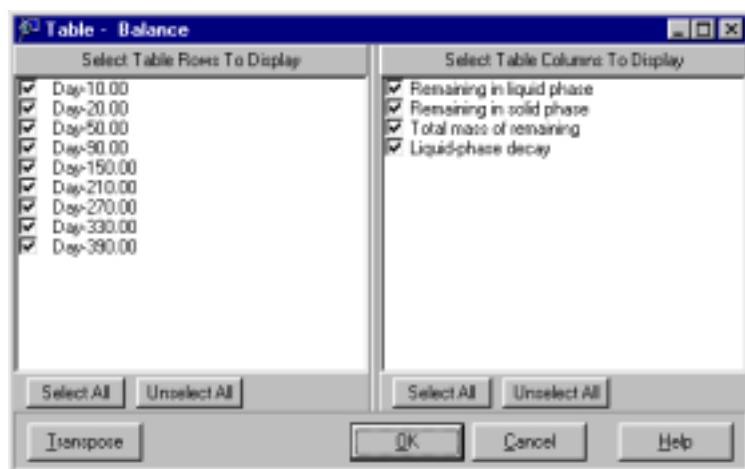
UnSat Suite allows you to view and edit PESTAN balance tables and add them to a report.

To access the desired table, click the arrow in the **Select Result Category** drop-down listbox and click **Tables**. In the lower **Select Tables Result at...** box you will get: **Balance**.



Click the icon to the right of the **Select Tables Result at...** box to add the output to the Output Tree.

In the Output Tree select the check box beside the **Balance** in the Output Tree to view all results available for this specific table. The following dialogue box will appear:



Here you may select desired output times and variables to customize your table. You may use the following tools for editing a table:

- ☞ **Unselect All** to unselect all times or variables and then click desired if you want to show only a small number of rows or columns in the table.
- ☞ **Select All** if you wish to specify all lists after you unselect some times or variables.
- ☞ **Transpose** if you want to switch columns and rows.
- ☞ **OK** after you have set a table.

The table will appear in the Result View:

Time	Depth	Value
Processing in local phase/Ensuring no valid phase/Total mass remaining (Lip)		
Day 0.00	2.0220E+06	2.0120E+06
Day 50.00	1.1990E+06	1.1900E+06
Day 100.00	0.1000E+06	0.1000E+06
Day 1000.00	4.4200E+06	1.0700E+06

You may change the size of the table fields by dragging boundaries of the field names and use the scroll-bar at the bottom of the table to view all results.

Creating a Report

To present results of your PESTAN simulation to your clients, you may use the UnSat Suite Report Generator.

To create a report and add the project input data, click the icon from the Operational Icons tool bar. The report will appear in a separate window.

By default, the Report Generator lists all input data for your project.

Parameter	Value	Units
Model Type	1	0.000000
Simulation Length	365.00	0.000000
Sampling Constant	0.000000	0.000000

Layer	Top (m)	Bottom (m)	Thickness (m)
PESTAN Waterloo Sand	0.0000	-10.0000	10.0000

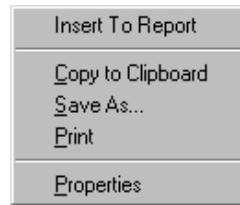
Parameter	Value	Units
Soil Density	1.6000	0.0000
Saturated Solute Content	0.0000	0.0000
Characteristic Curve Coefficient	0.0000	0.0000
Karstified Hydraulic Conductivity	0.0000	0.0000
Dispersion Coefficient	1.0000	0.0000
Fraction Organic Carbon	0.0000	0.0000

In the **Report** window you may edit the report, input your own text and add any type of graphics or table outputs produced by UnSat Suite.

Note: The graphs and tables will be placed at the insertion point.

To add a graph or table to the report:

- [1] In the **Report** window place the cursor in position where you would like your graph or table to appear in the report.
- [2] Create a graph or table using one of the methods described above and
- [3] <right click> in the Result View. The following menu will appear:



- [4] **Insert To Report.** The graph or table will appear in the report.

The graph may appear a size smaller than the original. To view the graph of desired size, click the graph in the **Report** window and stretch it until it reaches proper size.

A table may be longer than the Report window allows. In this case the table will be automatically wrapped.

Add necessary graphs and tables into the report and write your comments. You may insert a header and footer in your report, apply different fonts and styles while working in the **Report** window. To utilize these and other options, make corresponding selections from the top menu. After you are done, you may print the report or save it in your machine.

Part 4:

The VS2DT Model

Introduction

VS2DT (Variably Saturated 2-D Flow and Transport Model), is a well tested U.S. Geological Survey (USGS), finite difference model for cross-sectional, variably saturated flow and transport in porous media.

As part of UnSatSuite, VS2DT is a comprehensive 1-D unsaturated zone model based on the solution of Richard's equation. This model describes the transport of different types of contaminants (agricultural, industrial, (including radioactive)) with unsaturated water flow and its transformation in the soil and vadose zone and leaching of contaminant to groundwater.

Allowed profile structure: multilayer heterogeneous profile with an option to simulate hydraulic anisotropy.

Simulated processes:

Surface: surface storage (constant head), infiltration (constant flux), seasonal soil evaporation and plant transpiration, and vegetative growth.

Subsurface: unsaturated vertical flow (van Genuchten, Brooks and Corey and Haverkamp functions); pollutant transport with hydraulic dispersion, pollutant decay, pollutant adsorption (Freundlich and Langmuir isotherms), ion exchange (monovalent-monovalent, divalent-divalent, monovalent-divalent, divalent-monovalent).

Initial conditions may be inputted as moisture content or as pressure head for water and as concentration for chemical uniformly throughout the profile or by individual layers. An equilibrium profile may be specified above a user defined free water surface.

VS2DT is a most sophisticated unsaturated zone model that can serve as a primary standard for a solution of the most complicated practical problems. This model was used in different applications for solution of some important cases (e.g. wetland model). Until now it was not easy to use Windows interface in which you were able to utilize all potential capabilities of this powerful code.

UnSat Suite allows you to quickly prepare, run and interpret VS2D/T simulation by the use of graphical tools. The structure of the UnSat Suite interface allows you to decrease the number of input parameters by 2.5 times compared to the DOS version of the program.

VS2DT is equipped with a limited database of soil materials. In the next release of UnSat Suite, the database for several hundred soil types will be available.

10

Case Specifications

Case Settings

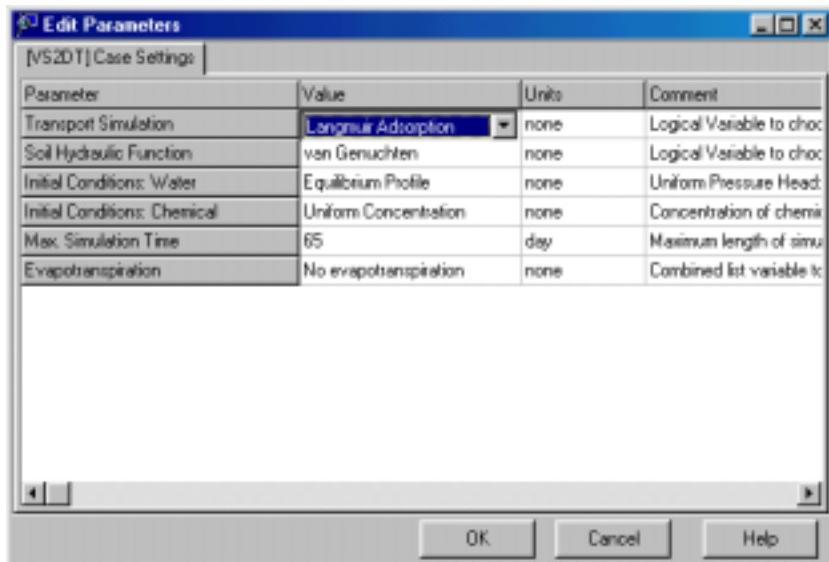
Constructing the profile requires the user to make several decisions regarding initial conditions and which types of processes will be simulated. These model options may be edited in the project parameter group labeled **Case Settings**. The various model options include:

- Transport Simulation Type;
- Soil Hydraulics;
- Initial Water Conditions;
- Initial Chemical Conditions;
- Simulation Time; and
- Evapotranspiration.

The following sections outline the various methods used for editing the case settings. Most case settings on selection introduce additional parameters. There are tables which outline the various selections possible for each case setting, the new location of each parameter that requires inputting after the selection was made, and the name of the new parameters that will become available with a one line description of the input parameter.

To edit all case settings:

- 1) a) <right click> on **Case Settings**.
b) Click **Edit**. The **Edit Parameters** window will open.
or
- 2) Double Click () on **Case Settings**.
- 3) Click an option for the appropriate parameter in its **Value** list.
- 4) The units are not editable. The default value **none** will remain.
- 5) Click [OK].



Transport Simulation

Transport Simulation is a logical variable that determines whether or not solute transport is simulated. Choices include: **No transport simulation, No Adsorption or Ion Exchange, Linear Adsorption, Freundlich Adsorption, Langmuir Adsorption, Monovalent-Monovalent, Divalent-Divalent, Monovalent-Divalent, and Divalent-Monovalent.**

The following chart contains the respective parameters that will become changeable in the **Transport Parameters** section for each soil depending on the chosen **Transport Simulation** method.

Selection	Location of Active Parameter	Active Parameter
No transport simulation	Each Soil's Transport Parameters	none
	Solver Settings	none
No adsorption or ion exchange	Each Soil's Transport Parameters	Default Parameters
	Solver Settings	Transport Equation Closure = Closure criteria for iterative solution of transport equation.
Linear Adsorption	Each Soil's Transport Parameters	Default Parameters K_d (Linear Adsorption) = Equilibrium distribution coefficient.

	Solver Settings	Transport Equation Closure = Closure criteria for iterative solution of transport equation.
Freundlich Adsorption	Each Soil's Transport Parameters	<p>Default Parameters</p> <p>K_f (Freundlich Isotherm) = Freundlich adsorption constant.</p> <p>n (Freundlich Isotherm) = Freundlich exponent.</p>
	Solver Settings	Transport Equation Closure = Closure criteria for iterative solution of transport equation.
Langmuir Adsorption	Each Soil's Transport Parameters	<p>Default Parameters</p> <p>K_1 (Langmuir Isotherm) = Langmuir adsorption constant.</p> <p>Q (Langmuir Isotherm) = Maximum number of adsorption sites.</p>
	Solver Settings	Transport Equation Closure = Closure criteria for iterative solution of transport equation.
Monovalent-Monovalent	Each Soil's Transport Parameters	<p>Default Parameters</p> <p>K_m (Ion exchange) = Ion exchange selectivity coefficient.</p> <p>Q_i (Ion exchange) = Ion exchange capacity of material.</p> <p>C_o (Ion exchange) = Total solution concentration for a pair of ions involved in the ion exchange process.</p>
	Solver Settings	Transport Equation Closure = Closure criteria for iterative solution of transport equation.
Divalent-Divalent	Each Soil's Transport Parameters	<p>Default Parameters</p> <p>K_m (Ion exchange) = Ion exchange selectivity coefficient.</p> <p>Q_i (Ion exchange) = Ion exchange capacity of material.</p> <p>C_o (Ion exchange) = Total solution concentration for a pair of ions involved in the ion exchange process.</p>
	Solver Settings	Transport Equation Closure = Closure criteria for iterative solution of transport equation.
Monovalent-Divalent	Each Soil's Transport Parameters	<p>Default Parameters</p> <p>K_m (Ion exchange) = Ion exchange selectivity coefficient.</p> <p>Q_i (Ion exchange) = Ion exchange capacity of material.</p> <p>C_o (Ion exchange) = Total solution concentration for a pair of ions involved in the ion exchange process.</p>
	Solver Settings	Transport Equation Closure = Closure criteria for iterative solution of transport equation.

Divalent-Monovalent	Each Soil's Transport Parameters	<p style="text-align: center;">Default Parameters</p> <p>K_m (Ion exchange) = Ion exchange selectivity coefficient.</p> <p>Q_i (Ion exchange) = Ion exchange capacity of material.</p> <p>C_o (Ion exchange) = Total solution concentration for a pair of ions involved in the ion exchange process.</p>
	Solver Settings	Transport Equation Closure = Closure criteria for iterative solution of transport equation.

The transport parameters section always contains the default parameters except when there is **no transport simulation**. The default parameters include:

Alpha L	Longitudinal dispersivity of the porous medium (can vary between 10cm - 300 cm for field site applications).
Alpha T	Transverse dispersivity of the porous medium.
D_m (Molecular Diffusion)	Molecular diffusion coefficient.
Decay Constant	Decay constant.
Dry Bulk Density	Dry Bulk density of solid phase.

Soil Hydraulic Function

Soil Hydraulic Function is a logical variable that defines the specific soil hydraulic function used for the simulation. Options include: **Brooks and Corey**, **van Genuchten**, and **Haverkamp**.

The following chart lists the active parameters found under **Soil Parameters** for each soil with their respective selection.

Selection	Active Parameters Under Each Soil's Soil Parameters
Brooks and Corey	H_b (Brooks and Corey) = Bubbling or air-entry pressure potential. Lambda (Brooks and Corey) = Pore size distribution index.
van Genuchten	Alpha' (van Genuchten) = van Genuchten's parameter for the pressure head-moisture content curve. Must be negative. Beta' (van Genuchten) = Exponent in van Genuchten's function for the pressure head-moisture content curve.

Haverkamp	A' (Haverkamp) = Pressure potential where relative hydraulic conductivity = 0.5. B' (Haverkamp) = Haverkamp's constant equal to the slope of the log-log plot of (1/Kr-1) versus the pressure potential. Alpha (Haverkamp) = Haverkamp's parameter for the pressure head-moisture content curve. Must be negative. Beta (Haverkamp) = Haverkamp's parameter for the pressure head-moisture content curve. Must be negative. Equal to slope of log-log plot of (1/S _e - 1) vs. h.
-----------	--

Initial Conditions

The initial water and chemical distributions may be set in the soil profile. Either as a uniform profile where all layers have the same conditions or non-uniformly where each layer may have a unique value. Depending on the selections made, the initial condition value must be entered under soil parameters or under profile initial conditions.

Initial Conditions: Water

Initial Conditions: Water indicates if initial conditions are read as pressure heads or moisture contents. Options include: **Uniform Moisture Content**, **Uniform Pressure Head**, **Equilibrium Profile**, **Nonuniform Moisture Content**, and **Nonuniform Pressure Head**.

The following chart lists the active parameters associated with each selection and where the active parameter(s) are located.

Selection	Location of New Parameter	New Parameter
Uniform Moisture Content	Profile Initial Conditions	Value of Moisture Content = Value of initial uniform moisture content.
Uniform Pressure Head	Profile Initial Conditions	Value of Pressure Head = Value of initial uniform pressure head.
Equilibrium Profile	Profile Initial Conditions	Minimum Head for Equilibrium Profile = Value of initial minimum pressure head for equilibrium profile. Groundwater Depth = Value of initial depth to ground water surface.
Nonuniform Moisture Content	Under each soil's Initial Conditions Parameters.	Value of Moisture Content = Value of moisture content for the layer at the beginning of simulation.
Nonuniform Pressure Head	Under each soil's Initial Conditions Parameters.	Value of Pressure Head = Value of pressure head for the layer at the beginning of simulation.

Initial Conditions: Chemical

Initial Conditions: Chemical is the chemical concentration at the beginning of the simulation. Options include: **Uniform Concentration**, **Nonuniform Concentration**.

The following chart lists the active parameters associated with each selection and where the active parameter(s) are located.

Selection	Location of Active Parameter	Active Parameter
Uniform Concentration	Profile Initial Conditions	Value of Initial Concentration = Value of initial uniform concentration through profile.
Nonuniform Concentration	Under each soil's Initial Conditions Parameters.	Value of Concentration = Value of chemical concentration for the layer at the beginning of simulation.

Maximum Simulation Time

The maximum simulation time determines the length of simulation, but does not activate any additional parameters regardless of the value inputted.

Max. Simulation Time Maximum length of simulation.

Evapotranspiration

The number of parameters required to be inputted by the user is dependent on the selected evapotranspiration setting. In addition, the parameters can be altered at different time steps to simulate different rates of evapotranspiration during different states in the simulation.

Evapotranspiration settings are very flexible. The user can choose to simulate either evaporation or transpiration, or both.

The following chart lists the active parameters associated with each selection and the location of the new parameter(s).

Selection	Location of Active Parameter	Active Parameter
Evaporation and transpiration	Evapotranspiration	Potential Evaporation Rate Surface Resistance to Evaporation Atmospheric Pressure Potential Potential Evapotranspiration Rate Root Depth Root Activity at Base Root Activity at Top Root Pressure
Evaporation only	Evapotranspiration	Potential Evaporation Rate Surface Resistance to Evaporation Atmospheric Pressure Potential

Selection	Location of Active Parameter	Active Parameter
Transpiration only	Evapotranspiration	Potential Evapotranspiration Rate Root Depth Root Activity at Base Root Activity at Top Root Pressure
No evaporation	Evapotranspiration	

Solver Settings

Solver settings allow the user to specify how the flow and transport equations are to be solved for each time step. The parameters that can be altered are:

- Maximum and minimum number of iterations per time step;
- Maximum number of time steps for the whole simulation;
- Relaxation parameter for how aggressive the solver converges on the solution;
- The equation solution accuracy or closure criteria;
- Calculation method for achieving the weighted hydraulic conductivity between cells (for all but a few cases, the geometric mean provides the most accurate values); and
- Space Differencing
- Time Differencing
- Display or not display Profile Balance after Every Time Step

These functions are pre-set to the most appropriate values and do not have to be mandatory specified for every new projects. However, if numeric solution will be unstable and the output variables will oscillate, correction of these functions is the only way to get a reliable result.

Editing the Solver Settings

You can edit the solver settings in two ways:



To edit the solver settings:

- 1) <right click> on **Solver Settings**.
- 2) Click **Edit**. The **Edit Parameters** window will open.
or **Solver Settings**
- 3) Type in the new value for the appropriate characteristic in its text box or click your selection from the **Value** list box.
- 4) Adjust the units in the **Units** list box.
- 5) Click **[OK]**.

Note: The Weighting Hydr. Cond., Space Differencing, and Time Differencing parameters Value boxes are drop-down lists. Select from the Value list to edit the parameter.

Note: The only parameters with changeable units are Flow Equation Closure and Transport Equation Closure.

The following parameters can be edited using the methods discussed above:

Flow Equation Closure

Closure criteria for the iterative solution of the flow equation (units used for head). A higher value will decrease the number of iterations required for head to converge.

Relaxation

Relaxation parameter for the iterative solving process. The relaxation

	parameter directly influences the time for convergence.
Transport Equation Closure	Closure criteria for the iterative solution of the transport equation (units used for concentration). Active only if Transport Simulation (Case Settings Group) is set not to No Transport Simulation. A higher value of this parameter will decrease the number of iterations required for concentration to converge.
Weighting Hydr. Cond.	Weighting option for inter-cell relative hydraulic conductivity. Should be 1 for full upstream weighting, 0.5 for arithmetic mean, 0.0 for geometric mean.
Min. Iterations	Minimum number of iterations per time step.
Max. Iterations	Maximum number of iterations per time step.
Space Differencing	Closure criteria for the iterative solution of the flow equation, units used for head. Choices include Center-in-Space or Backward-in-Space.
Time Differencing	Closure criteria for the iterative solution of the flow equation, units used for head. Choices include Center-in-Time or Backward-in-Time. We recommend to set this parameter to Backward-in-Time to avoid numeric oscillations. However, the solution will be less accurate than if Center-in-Time is used.
Max. Number of Time Steps	Maximum allowed number of time steps for the simulation.

For further explanations of the parameters, see the attached manual extract for the original VS2D/T model in the appendices of this manual.

Evapotranspiration Parameters

Evapotranspiration (ET) is the combination of evaporation and transpiration. There are four possibilities for simulating evapotranspiration. The user can select **No evapotranspiration**, **Evaporation and transpiration**, **Evaporation only** or **Transpiration**

only. With the **No evapotranspiration** option, no parameters need to be set in the evapotranspiration section.

Both evaporation and evapotranspiration are two stage processes. The first stage of the evaporation process begins with a wet soil surface. Liquid is leaving the soil system at a rate which is equal to the evaporative demand of the atmosphere. This continues until stage two. In stage two the water supply to the surface is insufficient to satisfy the demand of the atmosphere and the surface dries out. The **Potential Evaporation Rate**, the **Surface Resistance to Evaporation**, and the **Atmospheric Potential** all have to be inputted to simulate the evaporation process.

If the user chooses to model both transpiration and evaporation, the process is similar to evaporation. However, in evapotranspiration the soil surface supports vegetative growth. Therefore, the water supply from the soil is being evaporated into the atmosphere and it is also being extracted by roots growing in the soil to supply the water demands of the plants. The user must then input not only the evaporation parameters, but the evapotranspiration parameters as well. Such parameters include: the **Potential Evapotranspiration**, **Root Depth**, **Root Activity at the base and top** of the root zone, and the **Root Pressure**.

Editing the Evapotranspiration Parameters

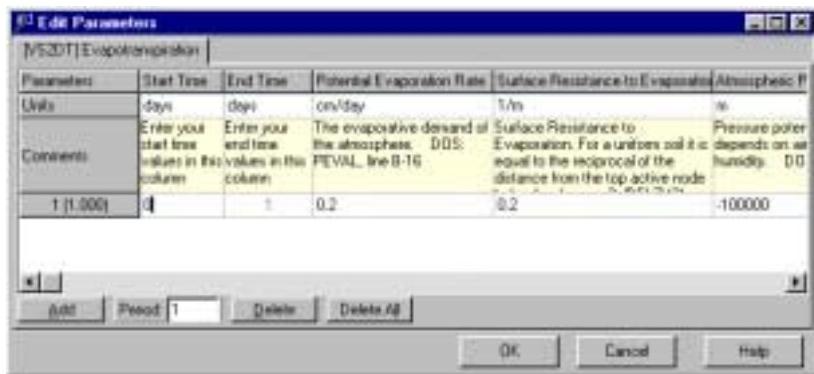
The **Evapotranspiration** parameters are subject to time steps with constant length. This allows you to accurately simulate effects due to climate changes over a long simulation time period.

By default the time steps are set at zero. You can add and delete time steps to suit the project needs.

To edit the Evapotranspiration parameters:

- 1) <right click> on **Evapotranspiration**.
- 2) Click **Edit**. The **Edit Parameters** window will open.
or  **Evapotranspiration** group name.

The **Edit Parameters** dialog box for evapotranspiration is shown below:



Time Steps

The length of the period to simulate the evapotranspiration is specified by the value entered by the user in the **Period** box. The **Start Time** and **End Time** fields show numbers for time steps but are not editable. The only value allowed to be edited in these fields is the start time of the first time step. Also for reference only, the grayed **Parameters** box displays the time step number and duration.

Time steps can be added by pressing [**Add**]. With each time step addition, the appropriate parameters may be altered.

As easy as time steps can be added, they can also be deleted.

To delete a time step:

- 1) Select the time step you wish to delete.
- 2) ⌘ [Delete].

All the time steps can be deleted by selecting [**Delete All**].

The simulation of the Evapotranspiration runs independently of the upper boundary condition for flow. However, it can be turned on or off by selecting the appropriate **Boundary Type** in the **Flow Upper Boundary** parameter group. This allows input of the evapotranspiration parameters according to the natural cycles of this process. The simulation of the Evapotranspiration runs at the background of the main simulation as a loop. After the last period of Evapotranspiration has been simulated, the first period starts again. This way of simulation reflects the cyclic nature of the evapotranspiration processes. For example, if the length of the simulation process is set to 30.4 days and the cycle includes 12 sets of values, the described subroutine will simulate the annual cycle of evapotranspiration.

Evapotranspiration Parameters

Potential Evaporation Rate Estimated evaporative demand of the atmosphere on the soil moisture.

Surface Resistance to Evaporation For a uniform soil it is equal to the reciprocal of the distance from the top active node to land surface.

Atmospheric Pressure Potential Depends on air temperature and air humidity, and under normal conditions varies between -100m and -2000m.

Potential Evapotranspiration Rate The estimated combined effects of evaporation (see above) and loss of moisture through plant uptake and subsequent release to the atmosphere (transpiration).

Root Depth Root depth at beginning of the ET period.

Root Activity at Base Root activity at base of root zone. Estimated as the length of roots in one unit volume of soil. Under normal conditions will vary between 0 and 3.0 (1/cm²).

Root Activity at Top Root activity at top of root zone. Estimated as the length of roots in one unit volume of soil. Under normal conditions will vary between 0 and 3.0 (1/cm²).

Root Pressure Pressure head in root at beginning of ET periods. Usually set to the permanent wilting point which is -150m for most agricultural crops.

See original VS2DT manual extracts for more explanation about parameters.

Boundary Conditions

Simulations with the VS2DT model require boundary conditions to be specified.

In VS2DT the upper boundary is always set at the soil surface while the location of the lower boundary may be specified at any point within the profile including points below the phreatic surface.

Two sets of parameters need to be specified at the upper and lower profile boundaries:

- flow conditions
- transport conditions

The following types of conditions are allowed at the profile **Flow Upper Boundary** which reflects interaction between the soil profile and the atmosphere:

- specified pressure head,
- specified flux,
- specified total head,
- evaporation,
- transpiration,
- evaporation and transpiration,
- and no specified boundary (impermeable boundary)

The following types of conditions are allowed for the profile **Flow Lower Boundary** which reflects interaction between the soil profile and the surrounding soil:

- specified pressure head,
- specified flux,
- specified total head,
- and no specified boundary (impermeable boundary)

The **Upper** and **Lower Transport Boundaries** do not differ in the type of condition they are able to simulate. For both boundaries the following types of conditions are allowed:

- specified concentration,
- specified mass flux,
- and no specified boundary

In VS2DT boundary conditions are time dependent. They have to be specified for the stress periods of variable length within which your knowledge of boundary processes allows you to presume uniformity. The length of stress periods is determined by the user. By managing types of boundary conditions and length of stress periods you may approximate any situation occurring at the upper and lower boundaries of the simulation domain. For recommendations on which type of boundary condition needs to be used to simulate specific practical case, look in the original VS2DT manual extract in the Manual folder of your installation CD-Rom.

The original DOS VS2DT model required the stress period to be of the same length for all boundary conditions. For example, if the Upper Flow Boundary is used to simulate daily changes in pressure head, all other boundary conditions must use the same stress periods. However, flow and transport processes at the boundary are independent by its nature. In UnSat Suite the length of stress periods for all four boundary conditions

may be specified independently which makes the data input process more efficient and the simulation of boundary processes more flexible.

Boundary Rules

Although the VS2DT model allows very flexible approximation of the boundary processes, not all combinations of flow and transport boundary conditions are allowed. The following chart is a summary of the permissible combinations for selecting boundary conditions.

Transport Boundary Conditions				
Flow Boundary Conditions	Fixed concentrations	Fixed mass flux	No specified boundary	Specified concentration of inflow
Fixed head, flow into domain	Permitted	Not Allowed	Permitted	Has to be Specified
Fixed head, flowhead out of domain	Not Allowed	Not Allowed	Permitted	Has to be Specified
Fixed flux into domain	Permitted	Not Allowed	Permitted	Has to be Specified
Fixed flux out of domain	Not Allowed	Not Allowed	Permitted	Has to be Specified
No specified boundary	Permitted	Permitted	Permitted	Not Required
Evaporation	Not Allowed	Not Allowed	Permitted	Not Required
Plant transpiration	Not Allowed	Not Allowed	Permitted	Not Required
Evaporation and transpiration	Not Allowed	Not Allowed	Permitted	Not Required

The execution of the boundary rules applies some restrictions on the combinations of the flow and transport boundaries. Under conditions when all boundaries are specified independently, it is hard to prevent the appearance of the forbidden combination during the user's input. The program allows the user to input any boundary type at any time. At a later stage the program recalculates the user-inputted stress periods into the model stress periods and checks the compatibility of boundary conditions. If a forbidden combination is detected, it will be replaced with the one allowed for the selected type of flow boundary. Fortunately, the most of forbidden combinations simulate very rare cases and the combinations which substitute them do not deteriorate the accuracy of results substantially.

Flow Boundaries

VS2DT Conventions

Pressure head is equal to zero at the phreatic surface or negative if the flow boundary simulates the unsaturated soil. Pressure head grows positively with a unit gradient below the phreatic surface.

Total head equals the sum of the pressure head and the elevation potential. The elevation potential is always negative within the profile because the datum is taken at the soil surface and the elevation is counted negatively downwards.

The flux into domain is counted positively and out of domain negatively.

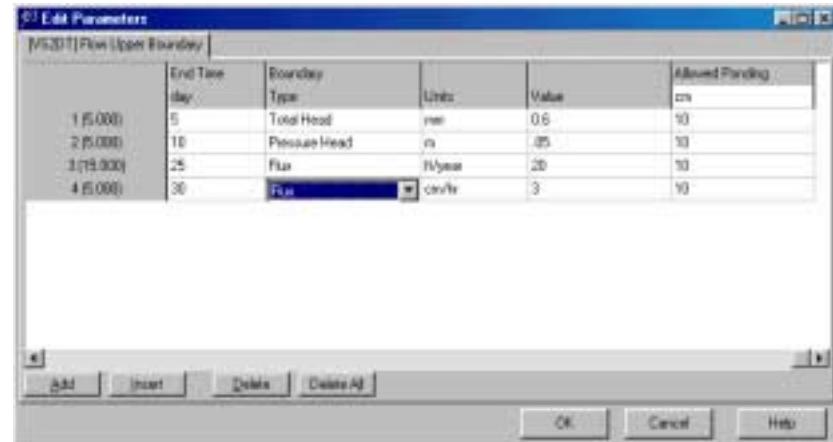
Setting Flow Boundaries

To edit flow boundaries, open the appropriate parameter group.

To open the Flow Boundary parameter group:

- 1) <right click> on the boundary you wish to edit.
- 2) Click **Edit**. The **Upper or Lower Flow Boundary** dialog box will open.
or $\text{F} \text{ } \text{F}$ the name of the group.

The **Edit Parameters** dialog box is shown below:



You can add time steps by $\text{F} \text{ } \text{F}$ [**Add**].

For each time step you can specify the **Start Time** and **End Time** in the appropriate boxes. The duration of the time step appears in brackets next to the time step number. Select the boundary condition for the time step from the **Boundary Type** list box. In the **Value** box, type the value of the flow, and select the appropriate units from the **Units** list box.

Ponding of water occurs when the rate of precipitation is greater than the rate of infiltration. The infiltration rate may be high at the beginning of the stress period, when the pressure head gradient in the soil is high but at a certain point the ability of soil to infiltrate water may get smaller than the intensity of flow at the boundary. As a result, the water accumulates above the surface of the profile. Therefore, ponding only occurs at the upper flow boundary, and is given as a height of water. The height is

assumed to be constant at all points on a flat surface. Any water that accumulates above this height is assumed to runoff/discharge to other areas and is not recharged into the profile. The units for ponding can be specified by selecting the appropriate units for the list box below the title column for Ponding. The value for ponding for each time step can be typed in the appropriate text boxes.

Time steps do not have to be added at the end. They can be added anywhere in the time period.

To insert a time step between two existing time steps:

- 1) Click the box below where your new step will be inserted.
- 2)  [Insert].

As easy as time steps can be added, they can also be deleted.

To delete a time step:

- 1) Click the box for the step you wish to delete.
- 2)  [Delete].

All the time steps can be deleted by  [Delete All].

Transport Boundaries

Transport boundaries can be set as either a specified concentration or as a fixed mass flux, or as no specified boundary. The specified concentration is the most commonly used type of transport boundary condition. For the boundary where flow is into the profile, the mass flux rate is calculated as the sum of influx rate times concentration of inflow plus the rate of dispersive flux from the boundary node.

The specified mass flux is used to represent a strictly diffusive flux and can only be combined with the impermeable flow boundary.

If **No Specified Boundary** condition is chosen for the boundary where flow is into domain, the mass flux rate is calculated as the sum of influx rate times concentration of inflow.

For stress periods or parts of stress periods when flow leaves the domain, no transport boundary conditions will be specified (See Boundary Rules). Under this condition the rate of solute flux out of the domain is equal to the rate of water flux times the concentration at the boundary node. Diffusive flux out of the domain is not allowed.

The evapotranspiration boundary condition assumes the water to be solute free.

Setting Transport Boundaries

The boundary condition can be altered at each time step to accurately represent the chemical's loading history at the boundary over a specified period of time.

To edit the flow boundaries, open the appropriate parameter group.

To open the Transport Boundary parameter group:

- 1) <right click> on the boundary you wish to edit.
- 2) Click **Edit**. The **Upper** or **Lower Transport Boundary** dialog box will open.
or $\text{F} \text{F}$ the name of the group.

The **Transport Boundary** dialog box is shown below:



You can add time steps by F [Add].

For each time step you can specify the **Start Time** and **End Time** in the appropriate boxes. The duration of the time step appears in brackets next to the time step number. Then, select the boundary type for the stress period from the **Boundary Type** list. Then, click the appropriate units from the **Units** list and type the desired value in the **Value** box. Input the value of **Inflow Concentration**, which will be used during the stress periods or parts of the stress periods when water enters the domain.

Time steps do not have to be added at the end. They can be added anywhere in the time period.

To insert a time step between two existing time steps:

- 1) Click the box below where your new step will be inserted.
- 2) F [Insert].

As easy as time steps can be added, they can also be deleted.

To delete a time step:

- 1) Click the box for the step you wish to delete.
- 2)  [Delete].

All the time steps can be deleted by  [Delete All].

Stress Period Defaults

Stress period defaults are default parameters used to set simulation for stress periods. In the stress period defaults section, you can specify the factors associated with time steps.

The length of the initial time step can be edited along with the maximum and minimum length of time steps to follow. You can also specify a multiplier for the initial time step length. Therefore, time steps will get larger or smaller depending on the nature of the simulation.

However, complications may arise during simulation. If convergence can not be achieved during a time step, a factor for time step reduction can be modified.

Specifying the Stress Period Defaults

The stress period default parameters can be edited using the following steps:

To access stress period default parameters:

- 1) <right click> on **Stress Period Defaults**.
- 2) Click **Edit**. The **Edit Parameters** dialog box will open.
or  **Stress Period Defaults**

The following dialog box will appear:



To edit stress period default parameters:

- 1) Select the appropriate units from the **Units** list where possible.
- 2) Edit the **Value** box.
- 3) Click **[OK]**.

The following stress period default parameters can be edited with the method mentioned above.

Initial Time Step	Length of an initial time step for the stress period.
Time Step Multiplier	Multiplier for the length of the time step for the stress period.
Maximum Time Step	Maximum length of time step for the stress period.
Minimum Time Step	Minimum length of time step for the stress period.
Reduction Factor	Factor by which time step length is reduced if convergence is not achieved.
Maximum Head Change	Maximum allowed change in head per time step.
Head Criterion	Steady-state head criterion to finish the iteration process for the time step. When the change in head between successive time steps is less than this value, it is assumed that steady state has been reached.

Setting Output Times

UnSat Suite allows the user to view model results for time steps. However, your desired output times may not coincide with the default time steps. In this case, you can specify your own output time.

Observation Times

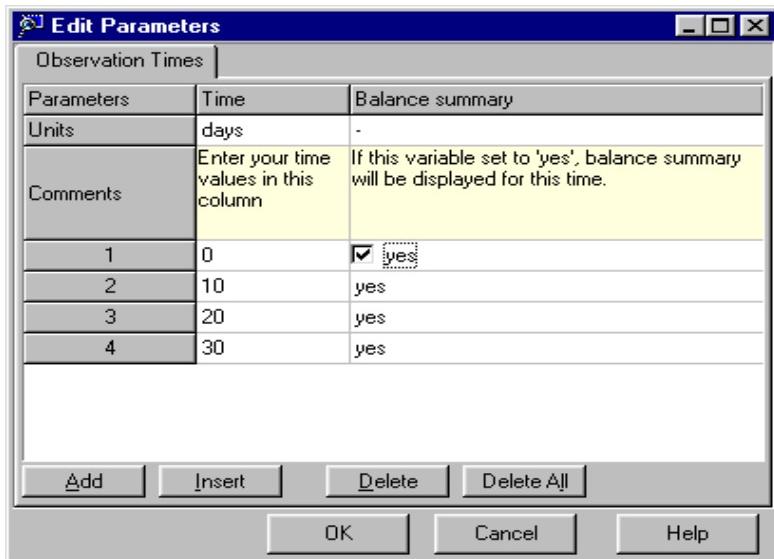
In the **Observation Times** dialogue box you have to input times to display results and choose if you are going to see results this model run. If the **Balance Summary** check box is set to **yes**, the balance summary for the selected time will be printed for the run. By default no time steps are set. You can add and delete time steps to suit your needs.

To open the **Observation Times** dialogue box, use one of the two following methods:

To open the Edit Parameters dialogue box:

- 1) <right click> on **Output Settings**.
- 2) Click **Edit**. The **Edit Parameters** dialogue box will open.
- 3) or <double click> on the **Observation Times**. The **Edit Parameters** dialogue box will open.

The **Observation Times** dialogue box is shown below:



To add observation times [Add]. When you click **Add** for the first time, the first line will be copied. Edit the value in the **Time** cell. Click **Add** and the new times will be added with the step equal to the difference between the first and second output times. If you want to change the time step for output times, edit the value of the last added time to bring the difference between the last two observation times to the desired value. When you click **Add**, the new times will be added with the new time step.

By default the value in the **Balance Summary** column is set to **yes**. If you do not need the output to be displayed at this time for the current run, click the cell and then click the appeared check box. The value will

change to **no**. Time steps do not have to be added at the end. They can be added anywhere in the time period.

To insert a time step between two existing time steps:

- 1) Click the boxes below where your new step will be inserted.
- 2)  **[Insert]**.
- 3) Change the time and rate.

As easy as time steps can be added, they can also be deleted.

To delete a time step:

- 1) Click the box for the step you wish to delete.
- 2)  **[Delete]**.

All the time steps can be deleted by  **[Delete All]**.

The Finite Difference Grid

VS2D/T uses the finite difference method to approximate the flow and transport the profile. To utilize this method, each layer must be separated into finite difference cells. The size of these cells can affect the convergence of the set of equations being solved. Furthermore, solution convergence is more difficult at the interface between layers in the profile where there may be large differences in the properties of the individual layers.

Solution convergence is improved by using reduced grid spacing near the outer boundaries or the boundaries of the profile layers. In the middle of the layer, the finite difference grid does not need to be as fine as at the boundaries. The grid design of fine grid spacing near the layer boundaries and larger grid spacing in the middle of the layer minimizes computational time and improves solution stability and time to convergence.

The UnSat Suite interface allows the user to set the grid quickly and effectively. Built in functions allow the user to customize parameters of the finite difference grid for each individual layer. These functions include:

Minimum Step length Set the minimum grid spacing.

Maximum Step length Set the maximum grid spacing.

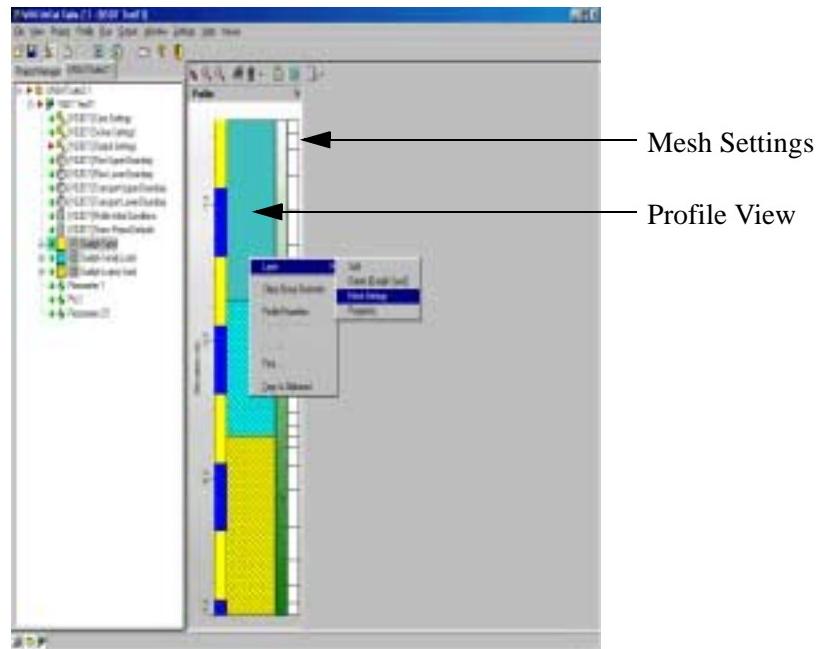
Multiplier	Increases the step interval by a constant multiplier through the layer until it reaches the Maximum Step length .
Symmetric	Creates a symmetric grid spacing where the grid is widest in the middle of the layer and narrowest at the boundaries.
Start at Bottom	Starts the finest grid spacing at the bottom of the layer.

Customizing the Finite Difference Grid

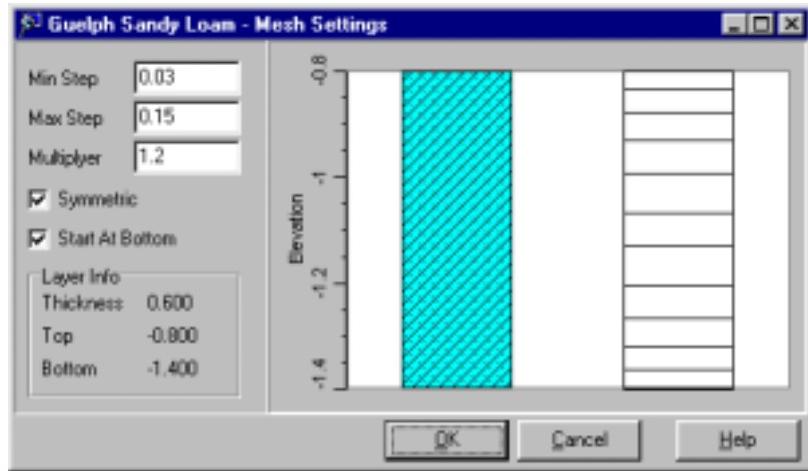
To set the grid characteristics:

<right click> on the layer to be edited in the Project Tree View and select **Mesh Settings**

or <right click> on the layer in the Profile View and select **Layer\Meshing Settings**.



The following dialog box will appear:



By default the following settings will be used for all profile layers:

Minimum Step length 1/20 of the layer thickness

Maximum Step length 1/4 of the layer thickness

Multiplier 1.2

If you clear the **Start At Bottom** button, the grid will start at the top with the coarsest grid spacing and gradually decrease in grid spacing down the layer. If you click the **Symmetric** button, the grid will be symmetric about the middle of the layer.

After you have edited the properties of the grid, click **[OK]** to save the changes.

Observation Points

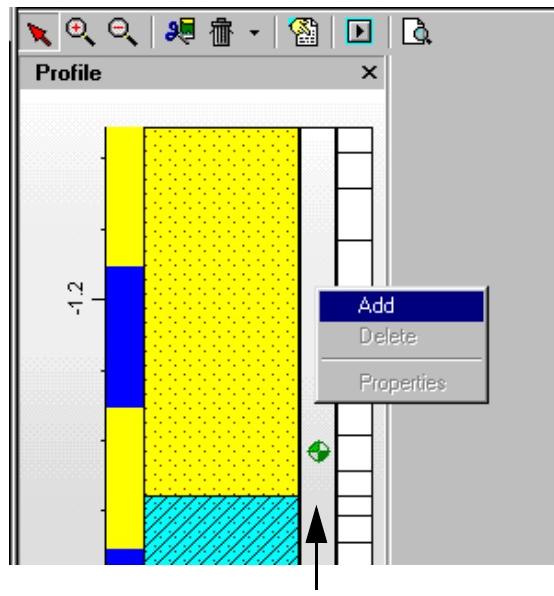
Model output may be saved separately at different cells along the soil profile by including observation points. Currently, observation points are restricted to the nodal values located at the centre of the finite difference cells. Any observation point depth may be entered into the profile; however, the observation point value will be obtained from the nearest node. Refining the grid spacing and dividing layers will allow greater flexibility in observation point locations. In future releases of UnSat Suite, a special interface will be developed to allow comparison of field observations to values calculated by the model for calibration.

Adding Observation Points

Please NOTE: At present, this feature is not enabled in VS2DT. The information contained below has been included as a reference for future releases of UnSat Suite.

To select the position of observation points:

<right click> on the **Observation Points Column** in Profile View.



Observation Points Column

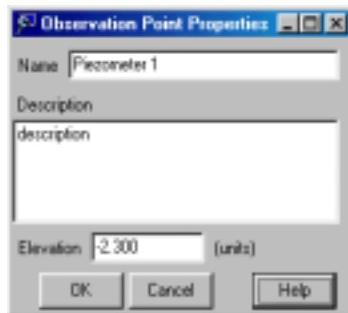
☞ **Add**

OR

<right click> on the profile name in the Project Tree View.

☞ **Add Observation Point**

The following dialog box will appear.



Enter the name in the **Name** text box and the depth where you wish to place your observation point, in the **Elevation** text box.

☞ **[OK]**

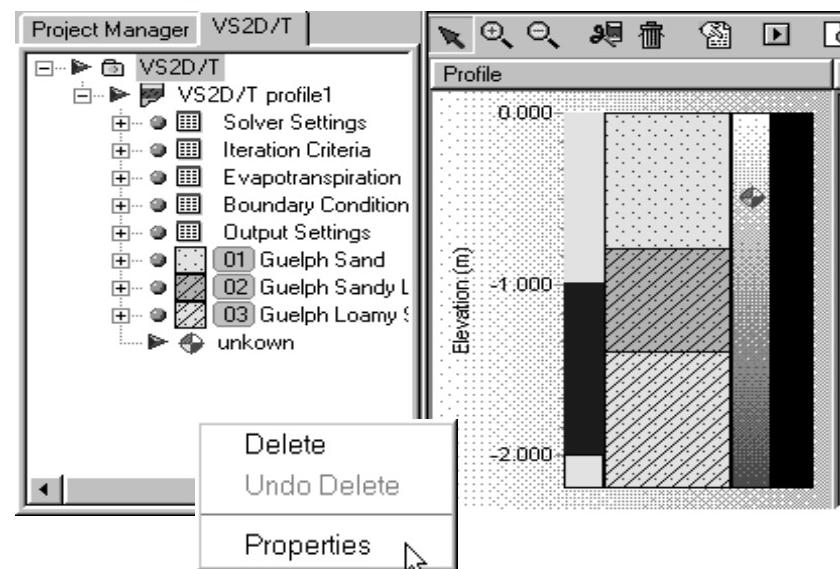
The observation point will appear in the Profile View and in the Project Tree View.

Observation Point Properties

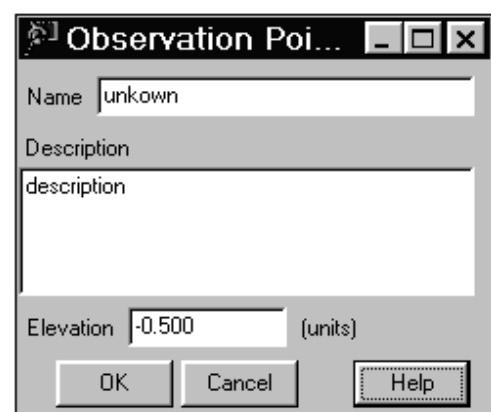
To change the properties of an observation point:

<double click> on the observation point in the Profile View.

Or <right click> on the observation point name in the Project Tree View and select **Properties**.



The following dialog box will appear.

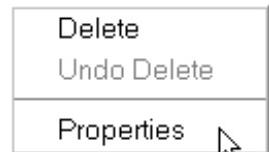


Edit properties of the observation point and [OK]

Deleting an Observation Point

To delete an observation point:

<right click> on the observation point in the Project Tree View.



☞ **Delete**

Restoring an Observation Point

To restore an observation point:

<right click> on the observation point in the Project Tree View.

☞ **Undo Delete**

Setting the Initial Conditions

When using VS2DT you must specify initial conditions for water and contaminant distribution. You may do it by layer or assign the values uniformly distributed within the profile.

Selecting the Type of Initial Condition

Water

To select the type of initial condition for water:

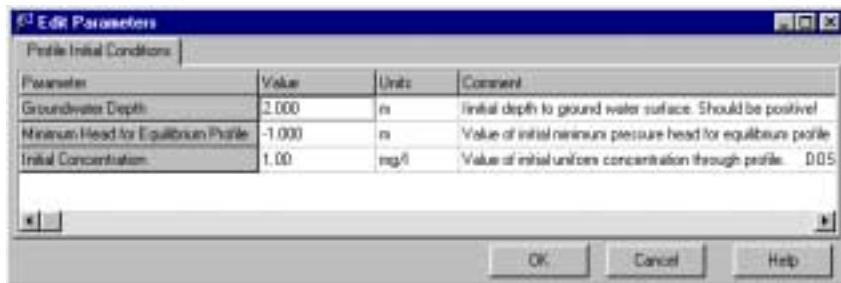
- [1] Open the **Case Settings** parameter group,
- [2] ☞ in the **Value** cell to the left of the **Initial Condition: Water**. The following list box will appear:



Select a type of initial condition that best represents actual site conditions and click **OK**.

If your choice is: **Uniform Moisture Content**, **Uniform Pressure Head**, or **Equilibrium Profile**, next you must specify appropriate parameters in the **Profile Initial Conditions** parameter group.

Profile Initial Conditions. The following dialog box will appear if **Equilibrium Profile** has been selected:



Enter the values for **Groundwater Depth** and **Minimum Head for Equilibrium Profile** here.

Concentration Distribution

To select the type of initial condition for chemical:

- [1] Open the **Case Settings** parameter group,
- [2] in the **Value** cell to the left of the **Initial Condition: Chemical**, the list box with the following choices will appear: **Uniform Concentration**, **Nonuniform Concentration**.

Make your selection to specify your initial conditions for chemical and click **OK**.

If your choice is: **Uniform Concentration**, next you must specify appropriate parameters in the **Profile Initial Conditions** parameter group as it was previously described.

Editing the Initial Conditions for a Layer

If in **Case Settings** parameter group you select **Nonuniform Moisture Content** or **Nonuniform Pressure Head for Water** and **Nonuniform Concentration for Chemical**, you must specify values for each layer separately.

You may access layer's initial conditions from the Project Tree View or Profile View.

To access Initial Conditions from the Project Tree View:

- 1) the **[+]** on the left side of the soil you are editing.
- 2) **Initial Conditions**

OR

- 1) the [+] on the left side of the soil you are editing.
- 2) <right click> on **Initial Conditions**.
- 3) Click **Edit**.

OR

- 1) <right click> on name of the soil you wish to edit.
- 2) Click **Properties**. The **Edit Properties** window will open.
- 3) Click the **Initial Conditions** tab.

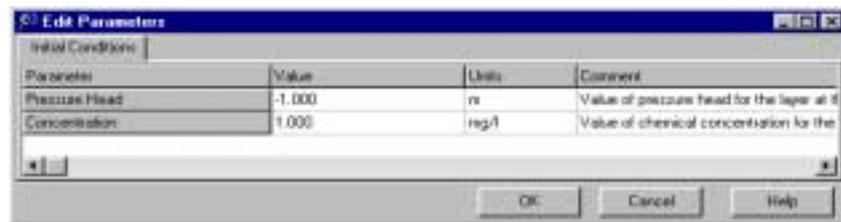
To access Initial Conditions from the Profile View:

- 1) <right click> on the layer
- 2) **Layer/Properties**. The **Profile Material Properties** window will open.
- 3) Click the **Initial Conditions** tab.

OR

- 1) on the layer. The **Profile Material Properties** window will open.
- 2) Click the **Initial Conditions** tab.

The dialog box will appear in result of the described actions:



Select units, if necessary, and enter the proper value for initial conditions in the **Value** cell.

OK to save changes.

11

Modifying the Profile

In this chapter, the tools for editing profile are explained. As was explained in “Profiles in UnSat Suite” on page 28, VS2DT works with natural profiles.

Applying VS2DT, the user can set the top and bottom elevation of the profile and edit them at any time during the input data preparation. However, editing of the layer thickness value is not allowed. To manipulate the soil layer structure within the fixed profile depth, the user has a set of graphical tools that can move the layer boundary, split and merge layers, and change the soil type within the layer. When the user moves the upper profile boundary graphically or changes the top profile elevation, the introduced changes will affect only the thickness of the upper layer. A move of the upper profile boundary or changes to the bottom profile elevation, will change only the thickness of the lower layer.

Profile Properties

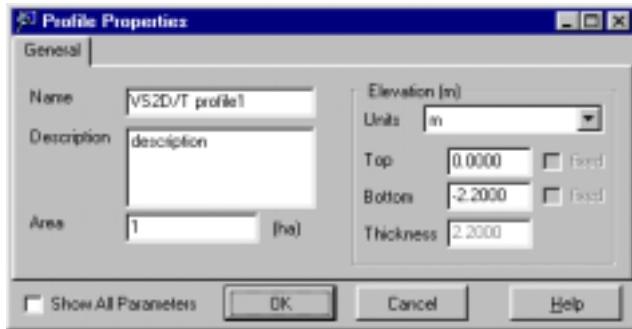
The basic profile properties include:

- Name,
- Description,
- Top and bottom elevation,
- Area,
- Units.

To edit the basic profile properties:

<right click> the profile in the Project Tree and click **Properties** or <right click> in the Profile View, and click **Profile Properties** or <double click> the profile name in the Project Tree View.

The **Profile Properties** dialog box will appear:



Name

A name of the profile.

Description

A description of the profile.

Area

A land area represented by profile (for information purpose).

Note: All balance constituents in VS2DT are calculated for a profile with the unit area 1 sq.cm. independently on the unit system selected. However, balance variables measured in volumes of water or pollutant mass are presented in the units selected by the user in the output.

Elevation: Top

The elevation of the uppermost layer of the profile. The uppermost soil layer will either stretch or contract depending on whether the top elevation will increase or decrease in result of editing. The minimum allowed top elevation is the lower boundary of the top soil layer.

Elevation: Bottom

The elevation of the lowermost layer of the profile. The bottom soil layer will either stretch or contract depending on whether the bottom elevation will increase or decrease in result of editing. The minimum bottom elevation is the upper boundary of the bottom soil layer.

Elevation: Units

Select the elevation length units from the units listbox.

Changing the Soil Profile Layer Structure

The layer structure of the default soil profile, and customized soil profiles, can be altered to correspond with a desired or observed soil profile. To maintain the profile's elevations and expedite changes, layer structure

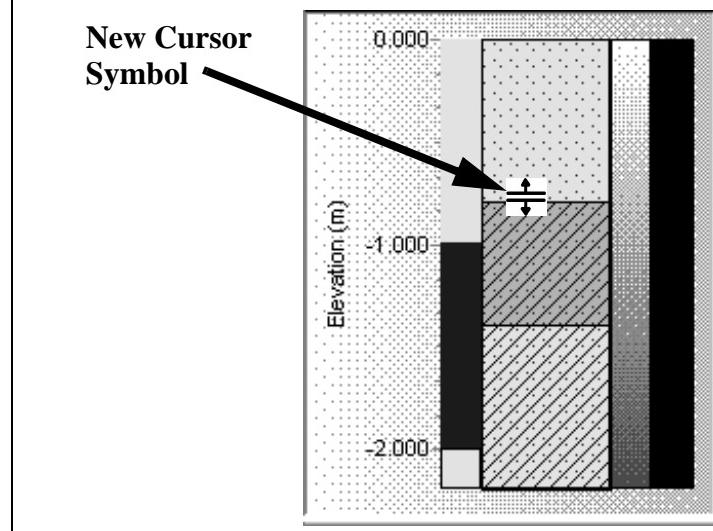
may be revised by either merging or splitting layers. Merging layers deletes one layer and expands the other layer into the space occupied by the deleted layer. Splitting layers divides a single layer into two layers. The properties for either layers may be changed to reflect the new desired profile.

Merging Layers and Erasing Layer Boundaries

A layer can be deleted by erasing its boundaries. When you erase a layer boundary the layer that shares the boundary takes over the span of the area formerly occupied by the erased layer. Therefore, the overall depth of the profile does not change.

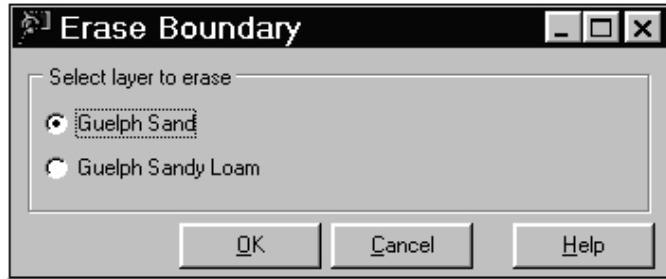
To erase layer boundaries:

- 1) Move the mouse arrow between two layers. The cursor symbol will change:



2) <right click>

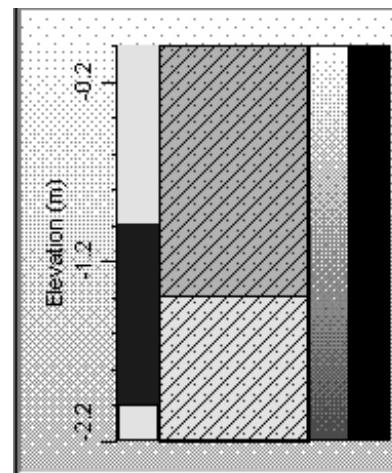
3) Click **Merge Layers**. The **Erase Boundary** dialog box will appear:



4) Select the option button for the layer you wish to erase.

5) Click **[OK]**.

The remaining layer will span the area once occupied by the deleted layer.



Restoring a Layer

To restore a layer:

<Right click> the layer's name in the Project Tree View.

2) Click **Restore**

3) 

Splitting a Layer

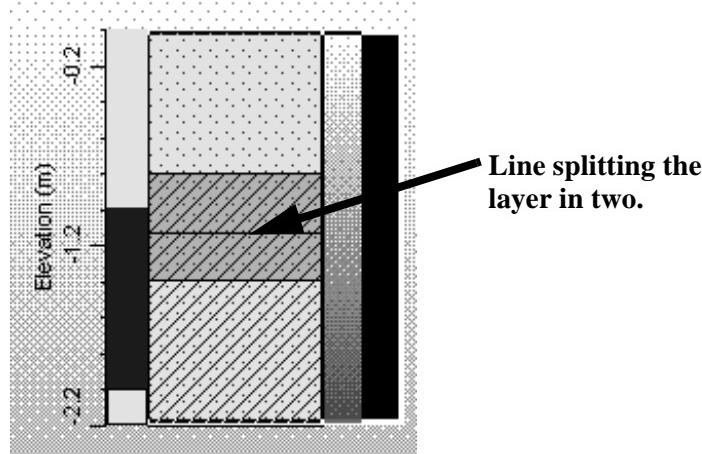
You can split a layer into multiple sections, and substitute material for each section. And you can assign different values for each parameter.

To split a layer:

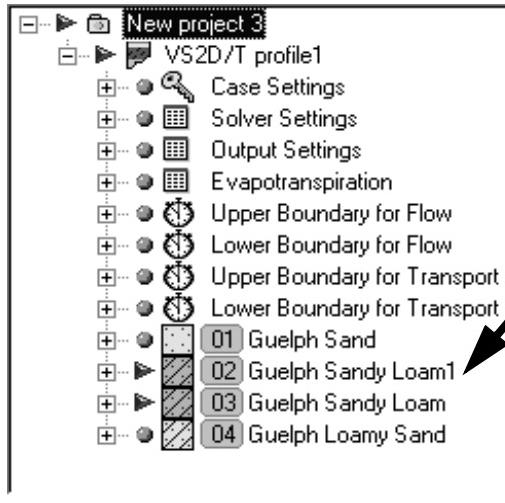
1) <right click> on the layer in the Profile View.

2)  **Layer/Split**

A line will appear through the layer at the mouse pointer position and a new layer will appear in the Project Tree.



Now each part of the layer can be edited separately with each section having its own unique properties. You may also substitute a material in the new layer.



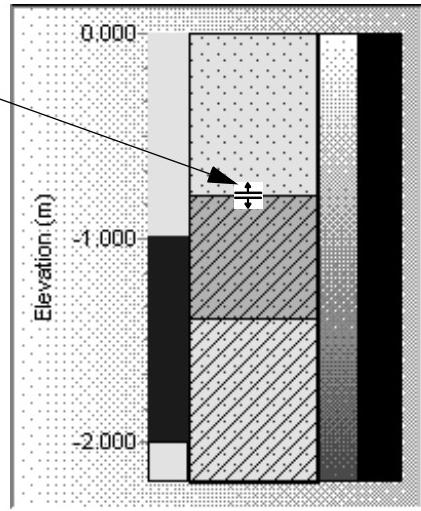
The two distinct sections of the layer.

Resizing a Layer

To resize a layer:

- 1) Move the mouse pointer between two layers. The pointer symbol will change.

New Cursor Symbol



- 2) Click and drag the boundary to its new location.
- 3) Either accept the new elevation or type the correct elevation in the **Confirm Value** dialog box.
- 4) Click [OK].

Changing Properties of a Layer

There are four different groups of parameters associated with a layer in VS2DT:

- General Parameters;
- Soil Parameters;
- Transport Parameters
- Initial Conditions.

To edit the properties of an existing layer in the profile, open the **Profile Material Properties** dialog box using one of the following methods:

double click the layer in the Profile View.

or double click the layer name in the Project Tree View

or <right click> the layer in the Project Tree View, and click **Properties**.

or <right click> the layer in the Profile View, and click **Layer\ Properties**.

A **Profile Material Properties** dialog box will appear.



To edit the property values, click the appropriate tab to open the parameter group that needs to be edited. Type your changes in the corresponding text boxes located beside each parameter. Edit the units of the parameters by selecting the appropriate unit from the associated list box. The changes will be saved by clicking the **[OK]** button.

Substituting Material in a Layer

To substitute material of a layer:

- 1) Open the **Profile Material Properties** dialog box.
- 2) Select, if necessary, new material category from the **Material Category** drop-down list box.
- 3) Click a new material from the **Material Category** list.
- 4) Give the new layer a unique name and write a descriptive comment.
- 5) Edit layer parameters or initial conditions if required.
- 10) Click **[OK]** to accept all changes.

Editing Soil Hydrologic Properties

Soil hydrologic properties determine conditions of water flow in the unsaturated zone. The salient featureS of the unsaturated zone is that both moisture content and the hydraulic conductivity of the material depends on the energy characteristic - pressure head. In VS2DT these dependencies can be approximated in three ways using the following functions:

- Brooks and Corey
- van Genuchten
- Haverkamp

Editing the Soil Parameters

You may access the soil properties from the Project Tree or from the Profile View.

To access soil properties from the Project Tree:

- 1) ↵ the [+] on the left side the soil parameters you are editing.
- 2) ↵ ↵ **Soil Parameters**

OR

- 1)  the [+] on the left side of the soil you are editing.
- 2) <right click> on **Soil Parameters**.
- 3) Click **Edit**. The **Profile Material Parameters** dialogue box will open.

OR

- 1) <right click> on name of the soil you wish to edit.
- 2) Click **Properties**. The **Edit Properties** dialogue box will open.
- 3) Click the **Soil Parameters** tab.

To access soil properties from the Profile View:

- 1) <right click> on the layer
- 2)  **Layer/Properties**. The **Profile Material Properties** dialogue box will open.
- 3) Click the **Soil Parameters** tab.

OR

- 1)  on the layer. The **Profile Material Properties** dialogue box will open.
- 2) Click the **Soil Parameters** tab.

General	Soil Parameters	Transport Parameters	Initial Conditions
Parameter	Value	Unit(s)	Comment
Anisotropy	1	-	Ratio of vertical-to-horizontal or radial conductivity [1]
Horizontal Sat.Hydr.Conductivity	100	cm/day	Horizontal permeability under the unit pressure gradient
Specific Storage	0.01	t/m³	Change in storage of the unit volume of saturated unit
Porosity	0.45	vol/vol	Total fraction of voids [DOS-HK (ITC-3), card B-7]
Gr	0.09	vol/vol	Residual moisture content [DOS-HK (ITC-5) (1.2)]
Alpha' (van Genuchten)	-70	cm	van Genuchten's parameter for the pressure head - n
Beta' (van Genuchten)	5	-	Exponent in van Genuchten's function for the pressure

In the dialogue box above you can edit the soil hydrologic parameters. Some of these parameters always exist in the dialog box, some of them appear depending on which choice was made in the **Soil Hydraulic Function** list box of the **Case Settings** parameter group.

Permanent Soil Hydrologic Parameters

The following parameters are not dependent on selections made in the **Case Settings** parameter group.

Anisotropy	Ratio of vertical-to-horizontal conductivity.
Horizontal Sat. Hydr. Conductivity	Horizontal permeability under the unit pressure gradient.
Specific Storage	Change in storage of the unit volume of material under the unit change of pressure.
Porosity	Total fraction of voids.
Q_r	Residual moisture content.

Dependent Soil Parameters

The remainder of the parameters are dependent on the choice of **Soil Hydraulic Function** selected. The following chart displays the options:

Selection	Parameters
Brooks and Corey	H _b (Brooks and Corey) = Bubbling or air-entry pressure potential. Lambda (Brooks and Corey) = Pore size distribution index.
van Genuchten	Alpha' (van Genuchten) = van Genuchten's parameter for the pressure head-moisture content curve. Must be negative. Beta' (van Genuchten) = Exponent in van Genuchten's function for the pressure head-moisture content curve.
Haverkamp	A' (Haverkamp) = Pressure potential where relative hydraulic conductivity = 0.5. B' (Haverkamp) = Haverkamp's constant equal to the slope of the log-log plot of (1/Kr-1) versus the pressure potential. Alpha (Haverkamp) = Haverkamp's parameter for the pressure head-moisture content curve. Must be negative. Beta (Haverkamp) = Haverkamp's parameter for the pressure head-moisture content curve. Must be negative.

For more explanations on parameters, see VS2DT original manual extract in the appendices.

Transport Parameters

Soil transport parameters determine conditions of chemical transport and transformation in the unsaturated zone. In VS2DT the processes describing chemical transport and transformation include:

- Advection
- Hydrodynamic Dispersion
- Decay
- Adsorption
- and Ion Exchange

The adsorption process can be approximated in three ways using the following empirical relationships:

- Linear Adsorption
- Freundlich Adsorption
- Langmuir Adsorption

The following processes of ion exchange can be simulated:

- Monovalent-Monovalent
- Divalent-Divalent
- Monovalent-Divalent
- and Divalent-Monovalent

Editing Transport Parameters

You may access the transport properties from the Project Tree View or from the Profile View.

To access Transport properties from the Project Tree View:

- 1) the [+] on the left of the soil you are editing.
- 2) **Transport Parameters**

OR

- 1) the [+] on the left side of the soil you are editing.
- 2) <right click> on **Transport Parameters**.
- 3) Click **Edit**. The **Edit Parameters/ Transport Parameters** dialogue box will open.

OR

- 1) <right click> on name of the soil you wish to edit.
- 2) Click **Properties**. The **Edit Properties** dialogue box will open.
- 3) Click the **Transport Parameters** tab.

To access Transport properties from the Profile View:

- 1) <right click> on the layer
 - 2) **Layer/Properties**. The **Profile Material Properties** dialogue box will open.
 - 3) Click the **Transport Parameters** tab.
- OR**
- 1) on the layer. The **Profile Material Properties** dialogue box will open.
 - 2) Click the **Transport Parameters** tab.

Note: The Transport Parameters group will appear for all profile layers provided the Transport Simulation selection (in the Case Settings group) is not set to No transport simulation.

The **Edit Parameters** dialogue box will appear as shown below.



Here you may edit transport parameters using the common tool of the UnSat Suite. The appearance of the **Transport Parameters** dialogue box depends on the selection made in the **Transport Simulation** listbox of the **Case Settings** parameter group. Some of the transport parameters always exist in the dialogue box, some of them appear depending on which choice was made in the **Transport Simulation** list box of the **Case Settings** parameter group.

Permanent Soil Hydrologic Parameters

The following parameters are not dependent on selections made in the **Case Settings** parameter group.

Alpha L	Longitudinal dispersivity of the porous media (can vary between 10cm - 300 cm for field site applications).
Alpha T	Transverse dispersivity of the porous media.
D_m (Molecular Diffusion)	Molecular diffusion coefficient.
Decay Constant	Decay constant.
Dry Bulk Density	Dry Bulk density of solid phase.

Dependent Transport Parameters

The following parameters appear depending on the selection made in the **Transport Simulation** list box of the **Case Settings** parameter group.

Selection made for Transport Simulation	Active Parameter
No adsorption or ion exchange	Default Parameters
Linear Adsorption	Default Parameters K_d (Linear Adsorption) = Equilibrium distribution coefficient.
Freundlich Adsorption	Default Parameters K_f (Freundlich Isotherm) = Freundlich adsorption constant. n (Freundlich Isotherm) = Freundlich exponent.
Langmuir Adsorption	Default Parameters K_1 (Langmuir Isotherm) = Langmuir adsorption constant. Q (Langmuir Isotherm) = Maximum number of adsorption sites.

Monovalent-Monovalent	<p>Default Parameters</p> <p>K_m (Ion exchange) = Ion exchange selectivity coefficient.</p> <p>Q_i (Ion exchange) = Ion exchange capacity of material.</p> <p>C_o (Ion exchange) = Total solution concentration for a pair of ions involved in the ion exchange process.</p>
Divalent-Divalent	<p>Default Parameters</p> <p>K_m (Ion exchange) = Ion exchange selectivity coefficient.</p> <p>Q_i (Ion exchange) = Ion exchange capacity of material.</p> <p>C_o (Ion exchange) = Total solution concentration for a pair of ions involved in the ion exchange process.</p>
Monovalent-Divalent	<p>Default Parameters</p> <p>K_m (Ion exchange) = Ion exchange selectivity coefficient.</p> <p>Q_i (Ion exchange) = Ion exchange capacity of material.</p> <p>C_o (Ion exchange) = Total solution concentration for a pair of ions involved in the ion exchange process.</p>
Divalent-Monovalent	<p>Default Parameters</p> <p>K_m (Ion exchange) = Ion exchange selectivity coefficient.</p> <p>Q_i (Ion exchange) = Ion exchange capacity of material.</p> <p>C_o (Ion exchange) = Total solution concentration for a pair of ions involved in the ion exchange process.</p>

For more explanations on parameters, see VS2DT's original manual extract in the appendices.

12

Running the Model, Viewing Output, and Reporting

Running the VS2DT Model

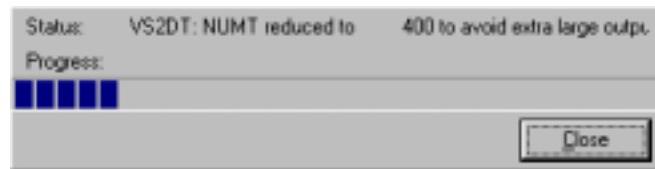


To run the model for a single profile, click the profile icon above the Profile View:



To run the model for multiple profiles or for one profile, if it is a single project, click the operational icon above the Project Tree View or **Run** in the main menu and then click **VS2DT**.

A progress bar will appear to indicate the computation progress, as shown below:



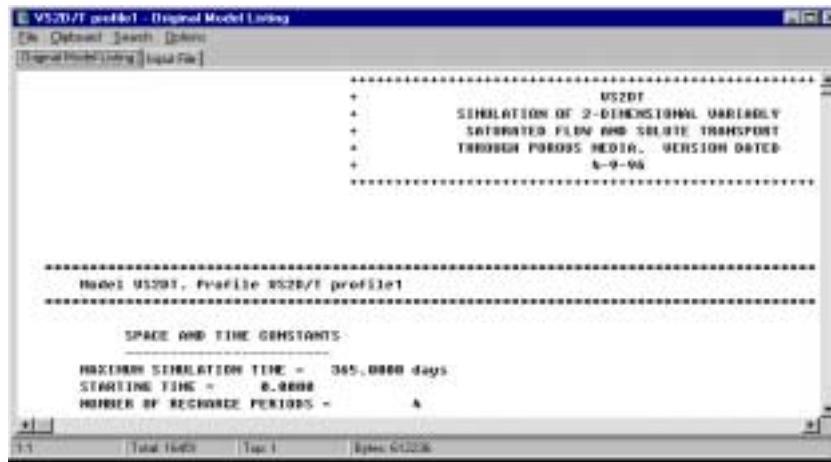
Viewing the Original VS2DT Input and Output Files

To view the original VS2DT output:

Output in the main menu, and then

View Original Listing

The following window will appear.

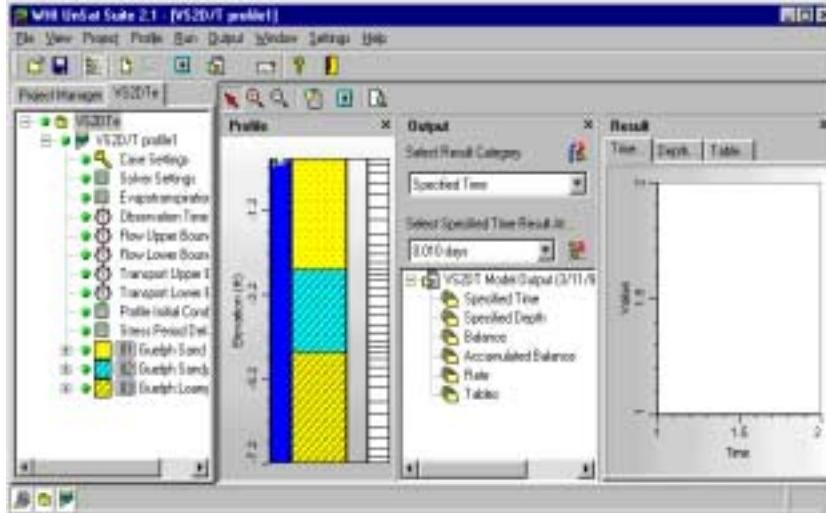


Here you can view the original listing, find specific expressions, mark and print parts or make a complete printout of the model results.

If you want to see the VS2DT input file, click the **Input File** tab.

Viewing the Output Graphs

After the model has successfully ran, the Output View and Result View window will open and the UnSat Suite window will appear as below:



 To enlarge the viewing area click the icon to close the Project Tree View, or click the 'X' in the Profile View to close it.

To select the output category under **Select Result Category**, select the type of output you want to view.

The first possible result group will appear in the list box below.



To view all available result groups, click the arrow in the **Select ‘Name of Category’ Result at...** drop-down list box.

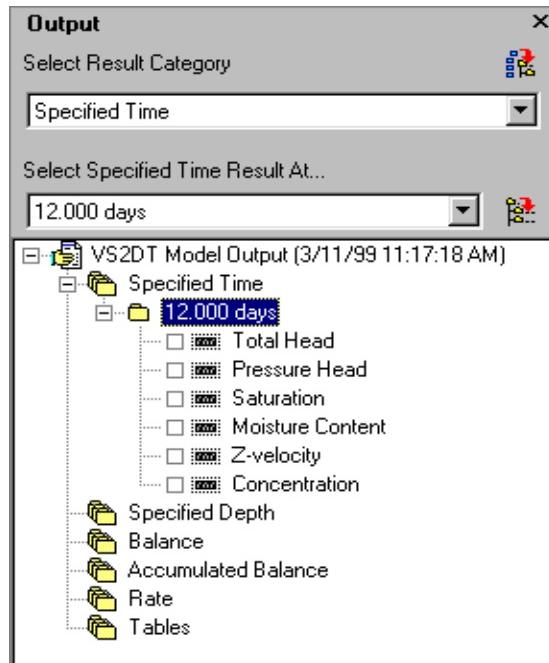
Specified Time and Depth

The list of observation times will appear if you select **Specified Time** in the **Select Result Category**. Under **Select Specified Time Result at...** choose the output time you wish to view.

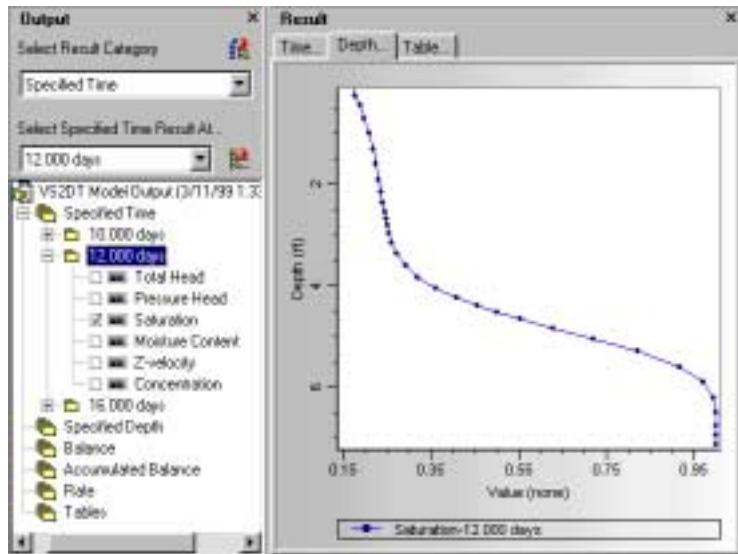


 To view all results available for this specific time, click the icon to the right of the **Select Specified Time Result at...** box.

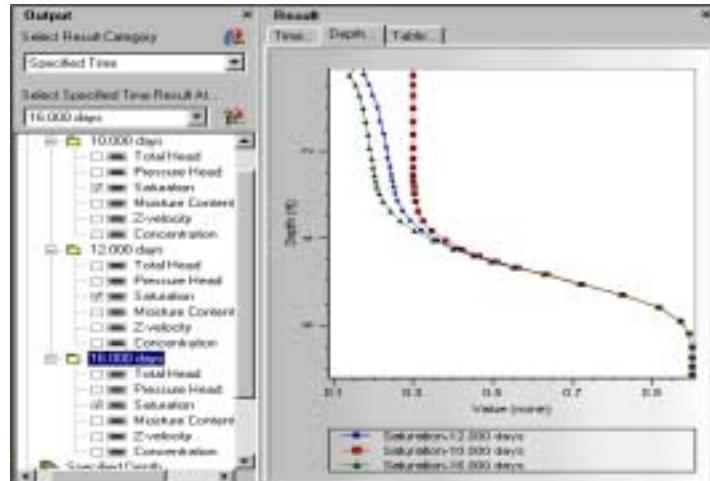
The list of available results will open in the Result Tree:



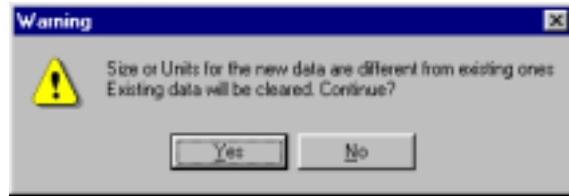
Click the check box beside the type of variable you wish to view. The graph of the variable will appear in the Result View window:



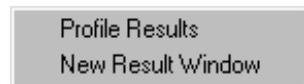
To add the graph for another times to the same window, select a new times from the **Select Specified Time Result At...** and check the same variables (you will get a warning if you choose different variables). The Result View will show profile distribution of the variable for different times:



If you wish to view output for other variables, click the corresponding check box. The warning will be posted if the new and previous variables are measured in different units:

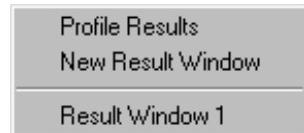


If you want to view both variables, you may place results for the second variable into the new Result Window. To do this, <right click> the name of the second variable. The following menu will appear:



Choose **New Result Window**. Results for the second variable will appear in the **Result Window 1**. You may add graphs for the other times to the **Result Window 1** using the same method.

If you want to see outputs for more than two variables, you may open additional Result Windows and place results there. To do this, <right click> the name of the additional variable. The following menu will appear:



Choose **New Result Window**. Results for the second variable will appear in the **Result Window 2**.

All the methods described for the **Specified Time** output viewing are applicable to the **Specified Depth** category. However, the graphs of this category will present time changes of the variable occurred at a specific depth.

Points of the profile depth are restricted to the nodes of the finite-difference mesh. The list of all nodes of the mesh will appear if you have selected **Specified Depth** category in the **Select Result Category** drop-down list box and click the drop-down arrow of the **Select Specified Depth Result at...** box:

2,400 cm
7,680 cm
14,016 cm
21,619 cm
30,743 cm
40,000 cm
49,257 cm
58,381 cm

Select desired depth from this list. Open the list of available variables and select variables to view, using the same tools as described earlier, for the **Specified Time** output category.

Balance, Accumulated Balance and Rate

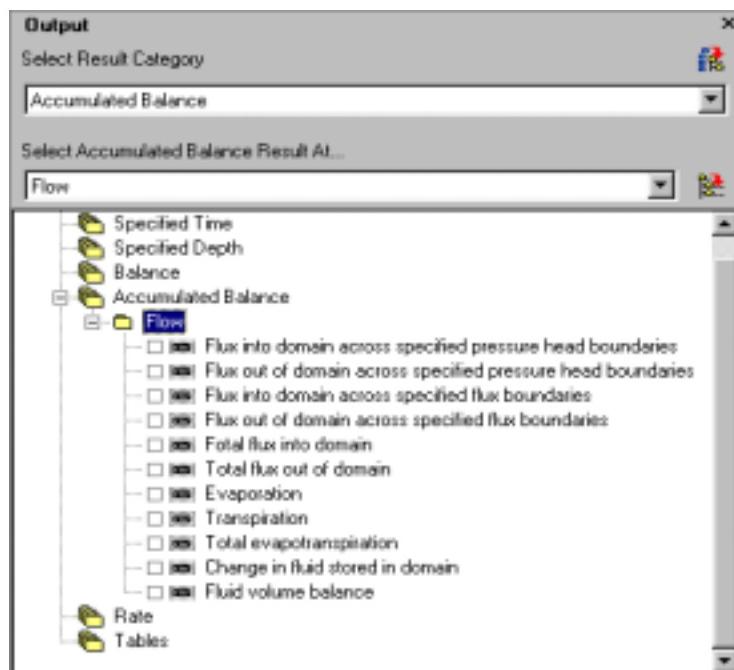
The VS2DT model allows you to view mass of the profile balance graphs and add them to a report.

To select the desired category, click the arrow in the **Select Result Category** drop-down list box and click **Balance, Accumulated Balance** or **Rate**. For each of the choices, you may select either **Flow** or **Mass** in the lower drop-down box of the Result View.



To view all results available for this specific type of output, click the icon to the right of the **Select Specified Time Result at...** box

The list of available variables will appear:



Select the desired variables and view them, using all tools described earlier.

Note: All balance constituents in VS2DT are calculated for a profile with the unit area 1 sq.cm. independently on the unit system selected. However, balance variables measured in volumes of water or pollutant mass are presented in the units selected by the user in the output.

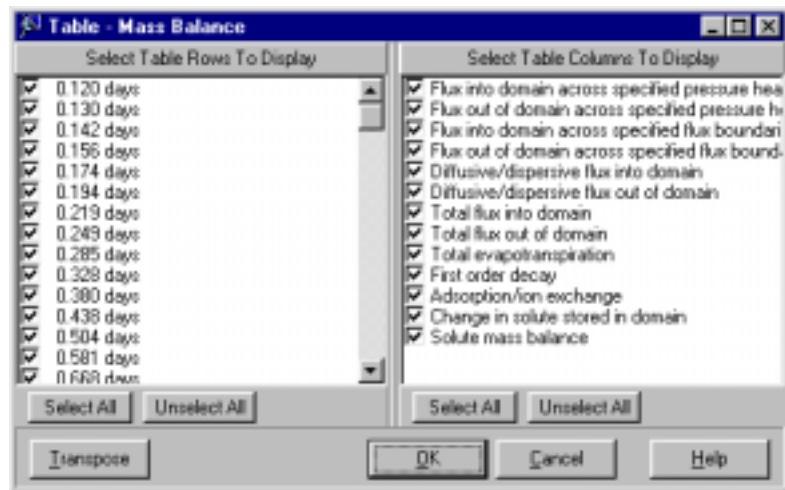
Viewing Tables

UnSat Suite allows you to view and edit VS2DT balance tables and add them to a report.

To access the desired table, click the arrow in the **Select Result Category** drop-down list box and click **Tables**. If you click the arrow in the lower drop-down box, you will get four types of tables to select:



Select the table from the list and click the icon to the right of the **Select Tables Result at...** box. The table will appear in the Output Tree. Click the check box beside the table in the Output Tree to view all results available for this specific table. The following dialogue box will appear:



Here you may select desired output times and variables to customize your table. You may use the following tools for editing a table:

☞ **Unselect All** to unselect all times and variables.

☞ **Select All** if you wish to specify all lists after you have unselected some times or variables.

☞ **Transpose** if you want to switch columns and rows.

☞ **OK** after you have set a table.

The following table will appear in the Result View:

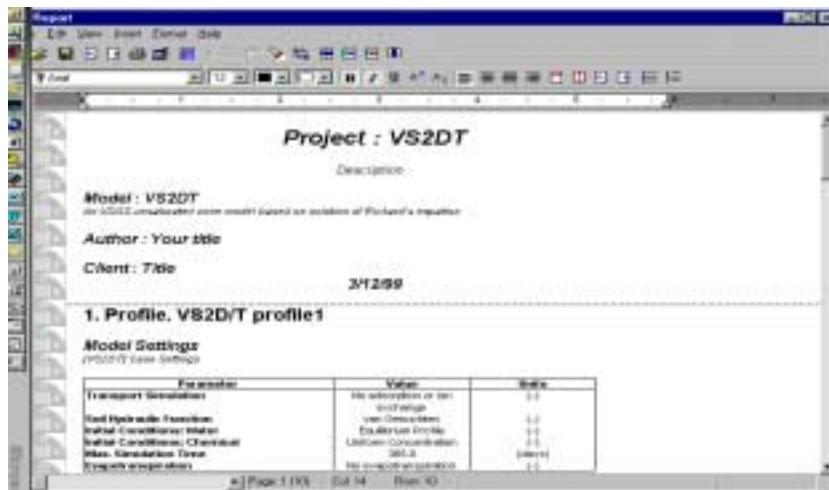
	Flow into domain (m/sec) ->	Flow out of domain (m/sec) ->	Flow into domain (m/sec) (<Flux out of	Flux out of domain (m/sec) (<Flux into domain (m/sec)
4,000 days	2.488E-01	-4.95E-01	0.000E+00	0.000E+00
8,000 days	2.627E-01	-2.707E-01	0.000E+00	0.000E+00
10,000 days	1.129E-01	-1.149E-01	0.000E+00	0.000E+00
12,000 days	0.000E+00	-2.042E-01	1.223E-02	0.000E+00
16,000 days	0.000E+00	-1.952E-02	2.983E-02	0.000E+00

You may change the size of the table fields by dragging boundaries of the field names and use the scroll bar at the bottom of the table to view all results.

Creating a Report

To present results of your VS2DT simulation to your clients you may use the UnSat Suite Report Generator.

To create a report and add the project input data, click the icon from the Operational Icons tool bar. The report will appear in a separate window:

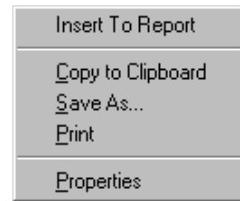


In this window you may edit the report, input your own text and add any types of graphics or table outputs produced by UnSat Suite.

Note: The graphs and tables will be placed at the insertion point.

To add a graph or a table to the report:

- [1] In the **Report** window place the cursor to position where you want your graph or table to appear in the report.
- [2] Create a graph or table using one of the methods described above.
- [3] <right click> in the Result View. The following menu will appear:



- [4] **Insert To Report.** The graph or table will appear in the report.

The graph may appear smaller than the original. To view the graph of desired size, click it in the **Report** window and stretch the graph until it reaches the proper size.

Editing Model Stress Periods

The running procedure for the VS2DT model differs slightly from the rest of the UnSat Suite models. The difference is you can review, and if appropriate, change settings for the **model stress periods**.

As explained before, the original DOS VS2DT model requires the stress period to be of the same length for all boundary conditions. However, flow and transport processes at the boundary are independent by its nature. In UnSat Suite the length of stress periods for all four boundary conditions may be specified independently, which makes the simulation of boundary processes more flexible. These stress periods are specified by the user for each boundary condition. After the data preparation process is finished and the user has saved the project settings, the user determined stress periods are calculated into the **model stress periods**. Each of the model stress periods is characterized by the unique combination of the boundary conditions. In addition to calculation of the user's defined stress periods, the interface also fills tail time gaps in the boundary condition schedule. If the total length of the period for which the boundary condition is determined is less than the total simulation length, the

remaining time will be filled with the last boundary condition stress period.

By default, all model stress periods are set with the default stress period parameters (specified in Stress Period Defaults parameter group). However, in many cases it may be necessary to edit these parameters for a particular model stress period. One such case is when the pressure head at the boundary changes rapidly and the model solution starts to oscillate. In this case, the reduction of the initial time step permits a solution to the problem.

To view or edit parameters of the model stress periods:

- ☞ **Run** in the main menu. The following list will appear.



- ☞ **Stress Periods.**

The **Stress Period Settings** dialogue box will appear:



Here you may edit parameters of the individual model stress periods. For more information on parameters, see “Stress Period Defaults” on page 192.

Part 5:

The VLEACH Model

Introduction

VLEACH is a popular US EPA model widely used for assessment of potential groundwater impacts and volatilization of volatile organic contaminants.

Allowed profile structure: one-layer homogeneous profile with initially uneven distribution of volatile contaminant.

Simulated processes:

Surface: constant recharge rate

Subsurface: constant flow and transport of volatile contaminant through soil, sorption, volatilization, diffusion to atmosphere and leaking of contaminant to groundwater

VLEACH is a 1-D finite difference model. The code simulates leaching in a soil polygon. The contaminant may be present in the soil as initial condition and may be introduced at the top boundary as a concentration for the recharge. A polygon is represented by a vertical stack of cells with constant depth that reach from the land surface to the groundwater table. The soil properties are considered to be uniform within the polygon. The initial contaminant concentration may vary from cell to cell.

The total mass of contaminant within each cell is partitioned among three phases: liquid (dissolved in water), vapor, and sorbed to solid surfaces. For simulation purposes, the total simulation time is divided into user-specified discrete time steps of constant length. During each time step there are three separate processes that take place. The contaminant in the liquid phase is subject to downward advection, and the contaminant in the vapor phase is subject to gas diffusion. Finally each cell is re-equilibrated according to the distribution coefficients. Gas diffusion can take place at the top and bottom boundaries. The mass flux in the liquid phase running across the bottom boundary is calculated. The model assume a steady-state downward water flow. The processes of in-situ degradation or production, and dispersion are neglected.

Assumptions incorporated in VLEACH do not allow it to simulate such sophisticated cases as VS2DT does. However, this model proved to be a very efficient tool for estimates of environmental impacts of industrial sources of groundwater contamination such as leaking from underground fuel tanks or spills from the pipelines. The ability of the model to account for volatilization and air diffusion of the volatile organic contaminants makes it a unique tool which only can be applied in many practical cases. This model has also a lot of advantages to be used for risk assessment studies which require multi-variant simulations.

UnSat Suite allows to quickly prepare, run and interpret VLEACH simulation by use of graphical tools. VLEACH is also equipped with a

limited database of soil materials and volatile organic contaminants. It is planned that in the next release of the UnSat Suite the database for several hundred of existing volatile organic contaminants will become available. It is also planned to develop a new feature which will allow running VLEACH for multiple polygons and integrate results for assessment of overall area-weighted groundwater impact for the entire area of the site.

13

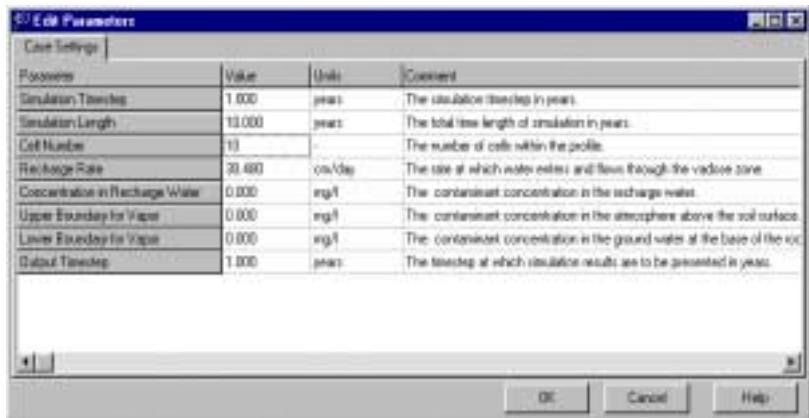
Input Specification

Specifying the Case Settings

The case settings section contains parameters which affect the entire profile.

To open the Case Settings parameter group:

- 1) the **Case Settings**,
- 2) Or <right click> the **Case Settings**.
- 3) Click **Edit**.



The following parameters can be edited in the **Case Settings** parameter group:

Simulation Length

The total time length of simulation.

Cell Number

The number of cells within the profile.

Recharge Rate

The rate at which water enters and flows through the vadose zone.

Concentration in Recharge Water	The contaminant concentration in the recharge water.
Upper Boundary for Vapor	The contaminant concentration in the atmosphere above the soil surface.
Lower Boundary for Vapor	The contaminant concentration in the groundwater at the base of the root zone.
Output Time Step	The time step at which simulation results are to be presented.
Simulation Time Step	The length of the simulation time step. Smaller Simulation Time Steps allow the model to more accurately determine the calculated results.

Specifying the Contaminant

Setting Initial Conditions

VLEACH allows you to specify the initial pollutant concentration in the soil within a single cell or a set of cells.

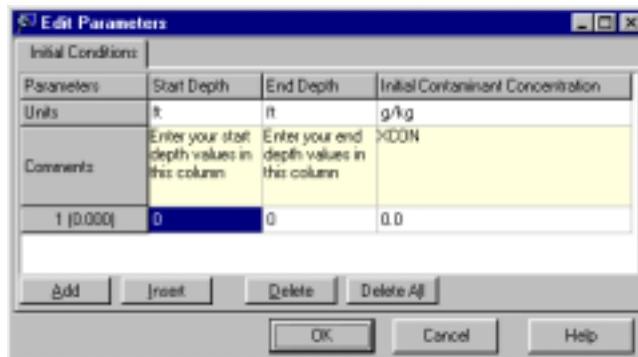
To set initial contaminant concentration:

<right click> on **Initial Conditions** in the Project Tree,

☞ **Edit**

Or

☞ ☞ on **Initial Conditions** in the Project Tree.



Enter the value of the **Start Depth** and **End Depth** for the first layer with uniform initial contaminant concentration. Change the units for **Initial Contaminant Concentration** if necessary, and enter the value of concentration for the layer.

[Add] to add additional depth increments. By default, the program will repeat the depth step for the previous layer and copy the value of the concentration.

Edit the start, end depths and concentration for the layers.

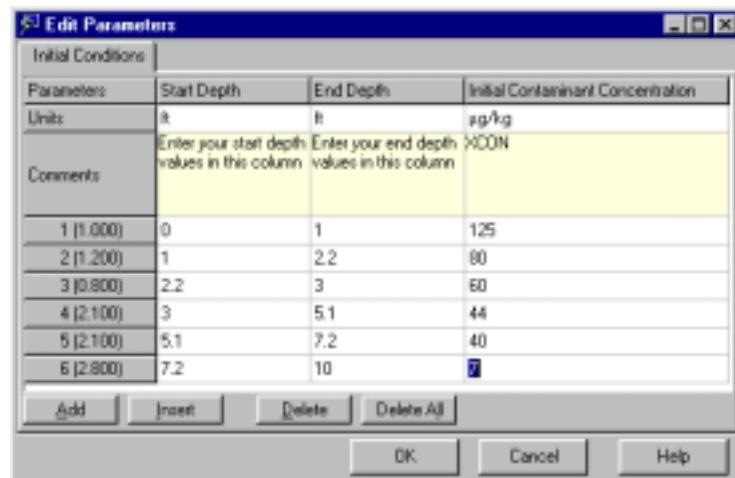
Although the VLEACH model requires the initial contaminant concentration to be input by cells that have a constant depth. You may input the initial contaminant concentration according to the observed distribution. The program will recalculate into cell concentrations, taking into account weights of different layers, if a cell includes more than one layer.

As easy as depth steps can be added, they can also be deleted.

To delete a depth increment:

- 1) Click the box for the step you wish to delete.
- 2) [Delete].

All the depth increments can be deleted by [Delete All].



[OK] after you are done.

Substituting the Contaminant

To substitute a chemical, open **Profile Material Properties** dialogue box for the chemical.

To open Profile Material Properties dialog box:

- 1) the **name** of the pesticide,
- Or
- 2) <right click> the **name** of the pesticide and click **Edit**.



the drop-down arrow of the **Material** list box. The list of available pesticides will appear:



Select a desired chemical and click it. After you have selected the chemical, you may edit its parameters to make them completely matching your case.

In UnSat Suite version 2.101 the list contains only three sample chemicals. It is planned to add several hundred common volatile chemicals to the database for the next version of this product.

Editing Contaminant Properties

You can access the properties of a contaminant in two ways:

To access contaminant properties:

- 1) the **name** of the pesticide,
 Chemical Parameters tab,
- Or
- 2) the **name** of the pesticide and click **Edit**,
 Chemical Parameters tab.

Chemical Parameters			
Parameter	Value	Unit	Comment
Water Solubility	1000.000	mg/l	Solubility of the chemical in water under the standard conditions
Organic Carbon Partition Coefficient	1000.000	ml/g	Characteristic of chemical describing the partitioning of the contaminant with organic carbon.
Henry Law Constant	0.010000		Characteristic of the liquid-gas partitioning of the contaminant.
Free Air Diffusion Coefficient	10000.000	cm ² /day	Characteristic of the transfer of the contaminant due to Brownian motion in the air phase.

The following contaminant parameters are changeable in this dialogue box:

Water Solubility

Solubility of the chemical in water under the standard conditions.

Organic Carbon Partition Coefficient Characteristics of chemicals describing the partitioning of the contaminant with organic carbon.

Henry's Law Constant

Characteristic of the liquid-gas partitioning the contaminant.

Free Air Diffusion Coefficient Characteristic transfer of the

contaminant due to Brownian motion in the air phase.

For more information on chemical parameters see extracts from the official VLEACH manual in the appendices.

Modifying the Profile

Profile Properties

To view or edit the profile properties:

<right click> the profile picture, and click **Profile Properties**,

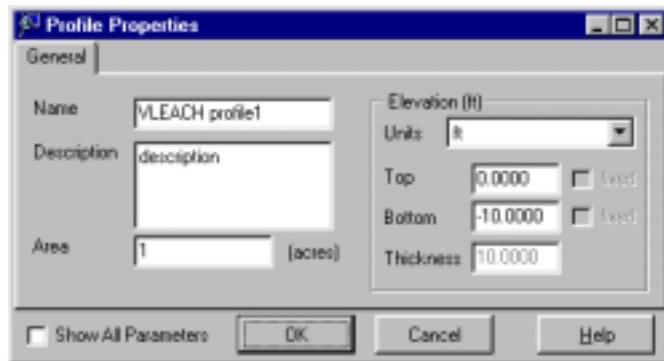
Or

 the profile name,

Or

<right click> the profile name and click **Profile Properties**.

The **Profile Properties** dialogue box will appear:



The following data are available in the **Profile Properties** dialogue box:

Name Type the name of the profile.

Description Type a description of the profile.

Elevation You may specify the elevation of the profile top or bottom.

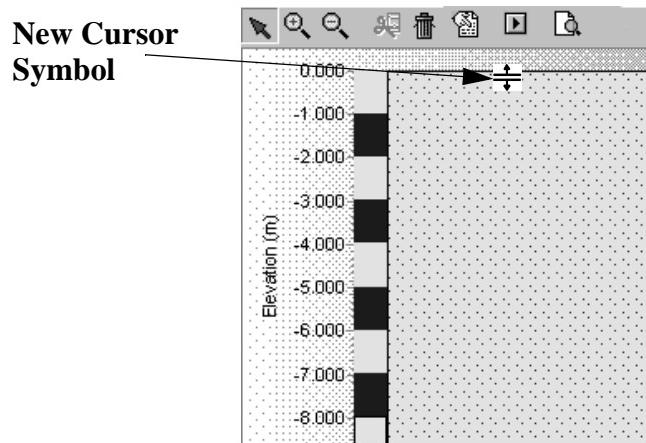
Area Type the land area represented by this profile.

Resizing the Layer

You may resize the layer through the **Profile Properties** dialogue box by changing its top and bottom elevation or graphically in the Profile View.

To resize a layer graphically:

- 1) Move the mouse arrow to the layer's boundary. The cursor symbol will change.

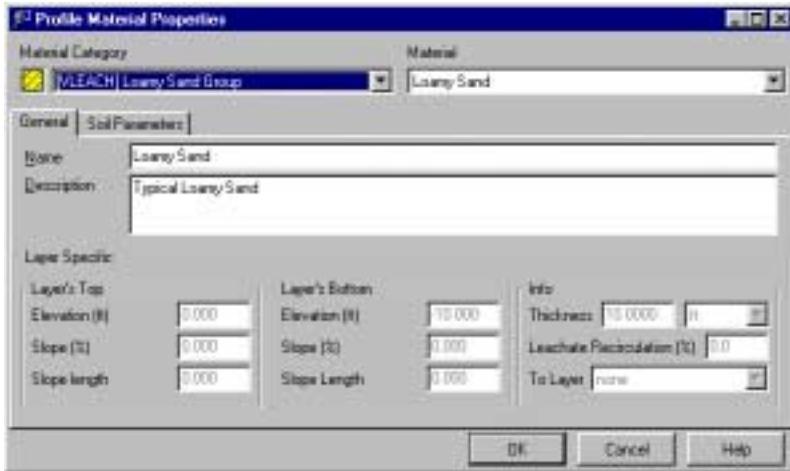


- 2) Click and drag the boundary to its new location.
- 3) Type the correct elevation in the **Confirm Value** dialogue box.

Substituting the Layer

To access a layer:

- 1) <right click> on the soil in the Profile View.
 - 2) Click **Layer/Properties**. The **Profile Material Properties** dialogue box for the soil will open,
- Or
- ☞ the layer name in the Project Tree View,
- Or
- <right click> the layer name and click **Properties**.



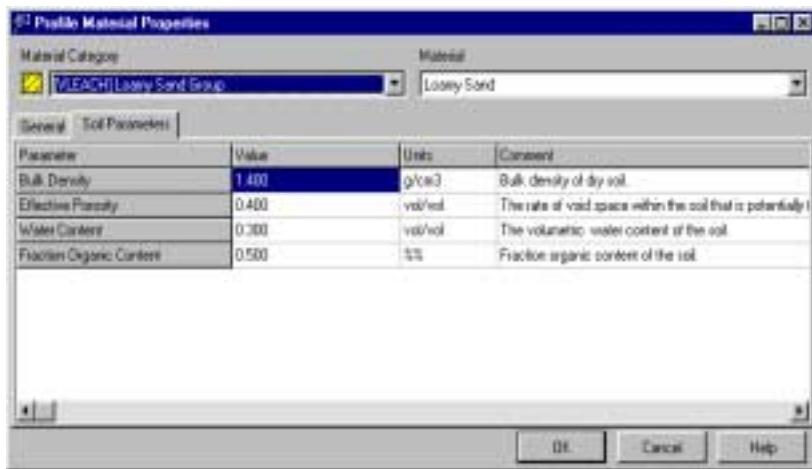
To substitute a layer:

- 1) Select a new material category from the **Material Category** list. The available soils will appear in the **Material** drop-down list box.
- 2) Select a new soil (material) from the **Material** list.
- 3) Give the new layer a unique name and write a descriptive comment.
- 4) Click the **Soil Parameters** tab and edit the values for soil parameters if necessary.
- 5) Click **[OK]**.

Editing Soil Properties

Open the **Profile Material Properties** dialogue box using one of the methods described in the previous section.

☞ the **Soil Parameters** tab.



The following are the changeable soil parameters:

Bulk Density

Bulk density of dry soil.

Effective Porosity

The rate of void space within the soil that is potentially fillable with water. The effective porosity equals total porosity minus irreducible water content. Since $P_{eff} = P_{tot} - WC$, and P_{eff} cannot be larger than P_{tot} (obviously), therefore WC must be less than or equal to P_{eff} .

Water Content

The volumetric water content of the soil.

Fraction Organic Content

Fraction organic content of the soil. You have two options for entering the F.O.C. The default units are % %, meaning that the value you have typed is a true percentage. Therefore, if you have 1% organics, type a value of 1 in this field, and use the % % units. The program will then convert this value into a decimal value. If you use the "part of unit" option for units, you would enter a decimal value of 0.01 to indicate 1% organics.

14

Viewing Output and Reporting

Original DOS VLEACH Output

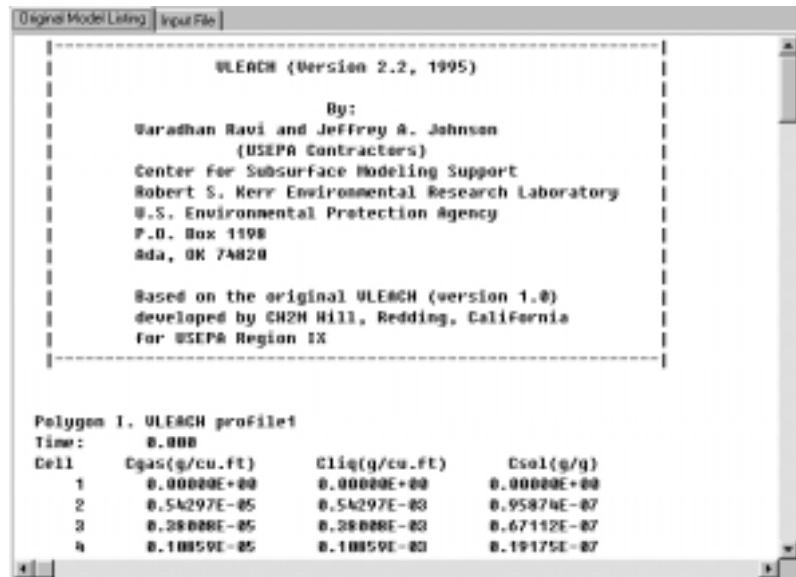
UnSat Suite allows you to view and print the original DOS VLEACH output.

To view and print original output:

☞ **Output** from the main menu.

☞ **Original Listing**

The **Original Model Listing** dialogue box will appear.



Scroll and view the original listing, find specific expressions, and print your results.

To view the VLEACH input file, click the **Input File** tab.

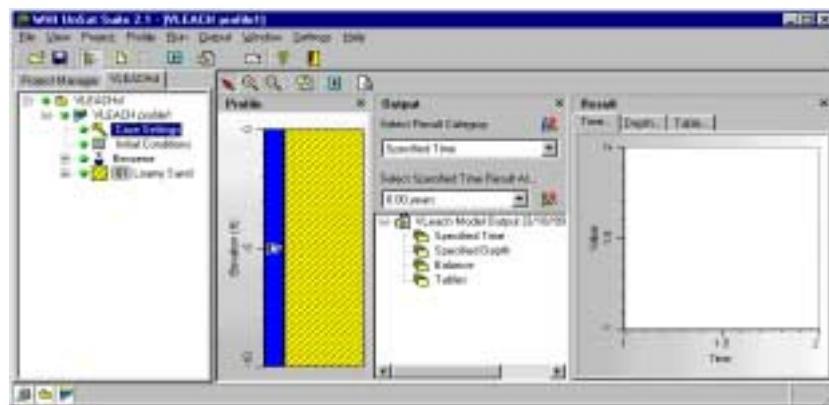
To print the file or the selected part of the file:

☞ **Print** from the **File** menu.

Specify the print properties, and click **[OK]** to print.

Viewing the Output Graphs

After the model has successfully ran, the Output View and Result View windows will open and the UnSat Suite window will appear shown below:



To enlarge the graph viewing area you may:



Click the icon to close the Project Tree View, or click the 'X' in the Profile View to close it.

To select the output category, click the arrow in the **Select Result Category** drop-down list box. The following list will appear:



Click the category you wish to view.

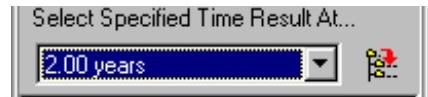
The first available result group for this category will appear in the listbox below. To view all available result groups, click the arrow in the **Select 'Name of Category' Result at...** drop-down listbox.

Specified Time and Depth

The list of output times will appear if you select **Specified Time** in the **Select Result Category** drop-down listbox:

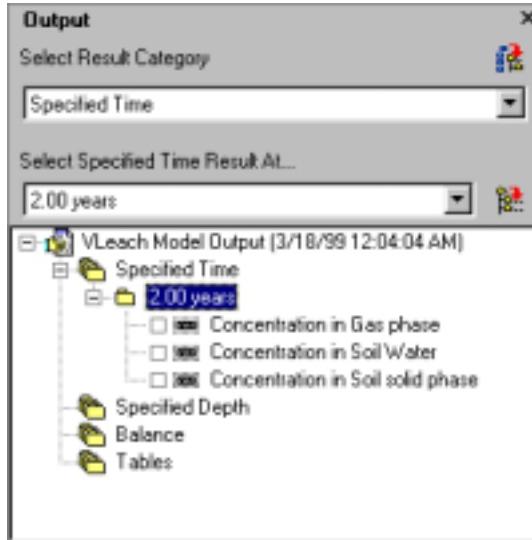


Use the slider to reach the time of your interest and click it. The selected time will appear in the drop-down box:



To view results for this specific time, click the icon to the right of the **Select Specified Time Result at...** box.

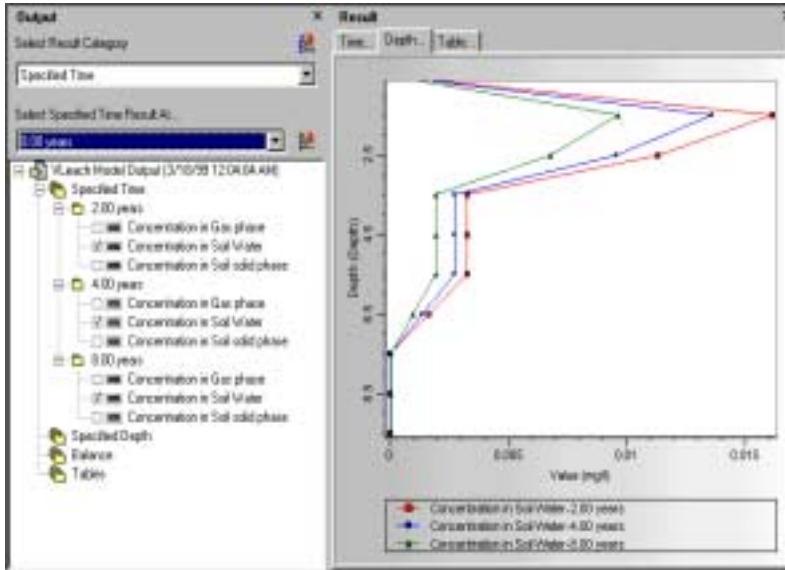
The list of results will open in the Result Tree:



Click the check box beside the type of variable you wish to view. The graph of the variable will appear in the Result View window.

To add the graph for another time to the same window, select a new time from the **Select Specified Time Result at...** box and check the same variables (a warning will display if you choose different variables). The

Result View will show profile distribution of the variable for different times:



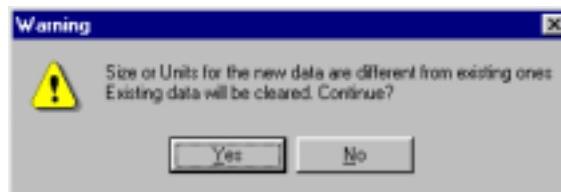
To erase an output for a specific time from the Result View window, unselect the corresponding check box in the Result Tree.

To clear the Result View window:

☞ **Output** from the main menu.

☞ **Clear Display Results**

If you wish to view output for other variable, click the corresponding check box. The warning will be posted if the new and previous variables are measured in different units:

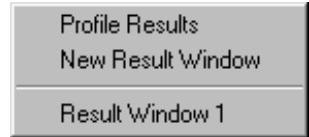


To view both variables which are measured in different units, you may place results for the second variable into the new Result Window. To do this, <right click> the name of the second variable. The following menu will appear:

Profile Results
New Result Window

Choose **New Result Window**. Results for the second variable will appear in the **Result Window 1**. You may add graphs for the other times to the **Result Window 1** using the same method.

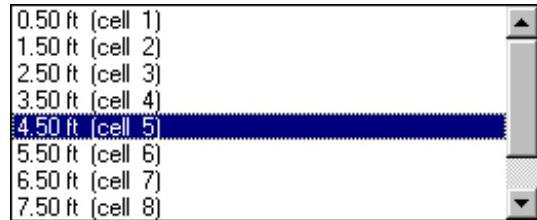
To see outputs for more than two variables, you may open additional Result Windows and place results there. To do this, <right click> the name of the additional variable. The following menu will appear:



Choose **New Result Window**. Results for the second variable will appear in the **Result Window 2**.

All the methods described for the **Specified Time** output viewing are applicable to the **Specified Depth** category. The graphs of this category will present time changes of the variable occurred at a specific depth.

The output for specified depth is produced for all cells. The list of cells will appear if you click the drop-down arrow of the **Select Specified Depth Result at...** box:



Select desired depth from this list, open the list of available variables and select variables to view using the same tools which were described before for the **Specified Time** output category.

Balance

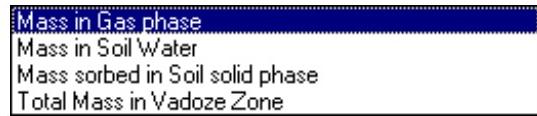
The VLEACH model allows you to compute the balance of the profile. In UnSat Suite you may view the balance graphs and add them to a report.

Select Balance in the **Select Result Category** drop-down listbox.

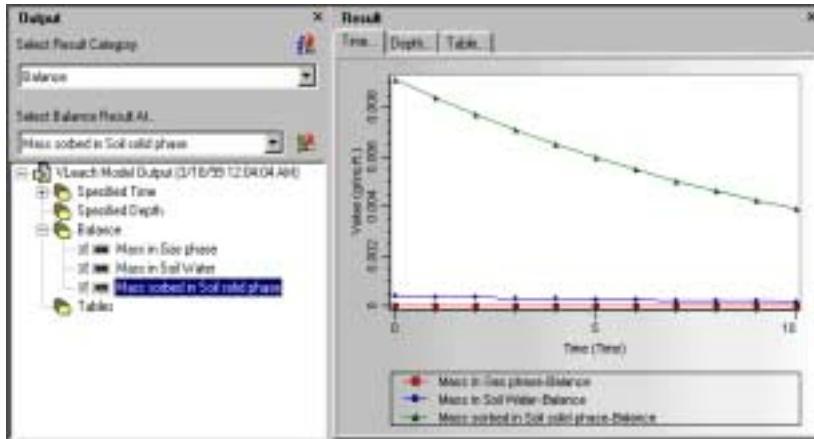


To view all results available for this specific type of output, click the icon to the right of the **Select Balance Result at...** box:

The list of available variables will appear:



Select the desired variables and view them using all tools described earlier.



Viewing Tables

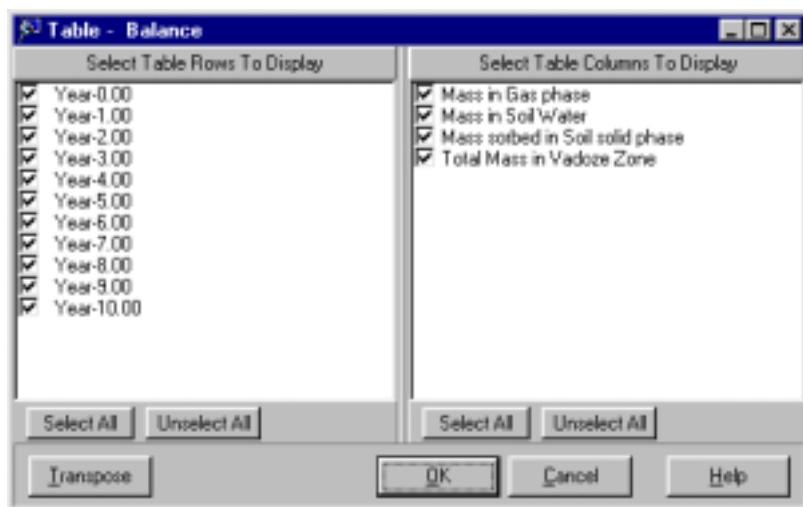
UnSat Suite allows you to view and edit VLEACH balance tables and add them to a report.

To access the desired table, click the arrow in the **Select Result Category** drop-down listbox and click **Tables**. In the lower **Select Tables Result at...** box you will get: **Balance** and **Accumulated Balance**.



Click the icon to the right of the **Select Tables Result at...** box to add the output to the Output Tree:

In the Output Tree select the check box beside the **Balance** to view all results available for this specific table. The following dialogue box will appear:



Here you may select desired output times and variables to customize your table. You may use the following tools for editing a table:

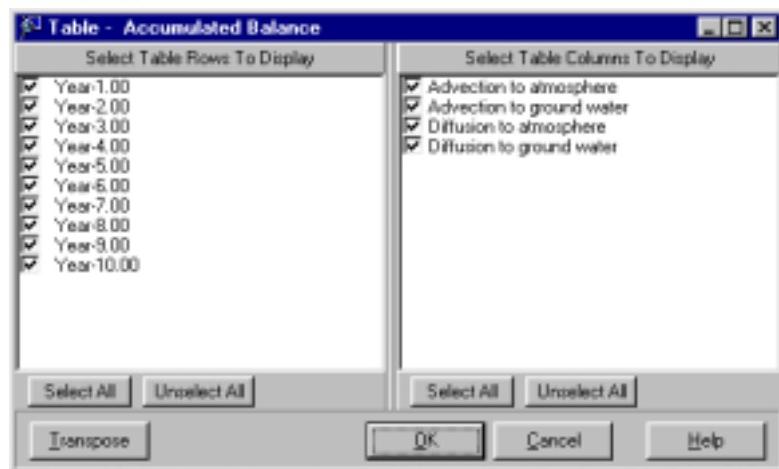
- ☞ **Unselect All** to unselect all times or variables and then click desired if you want to show only a small number of rows or columns in the table.
- ☞ **Select All** to specify all lists after you unselected some times or variables.
- ☞ **Transpose** to switch columns and rows.
- ☞ **OK** after you have set a table.

The table will appear in the Result View:

	Mass in Gas phase (kg/m ³)	Mass in Soil Water (kg/m ³)	Mass in soil solution (kg/m ³)
Year 0.00	1.2031E-06	2.3094E-04	3.1216E-03
Year 1.00	1.1872E-06	2.5917E-04	8.3988E-03
Year 2.00	1.0890E-06	2.2068E-04	7.6986E-03
Year 3.00	1.0709E-06	3.8371E-04	7.0739E-03
Year 4.00	8.2842E-07	2.7053E-04	6.4880E-03
Year 5.00	8.5297E-07	2.5595E-04	5.9709E-03
Year 6.00	7.0305E-07	2.3513E-04	5.5856E-03
Year 7.00	7.1397E-07	2.5598E-04	5.0298E-03
Year 8.00	6.6149E-07	1.8664E-04	4.6202E-03
Year 9.00	6.0370E-07	1.8271E-04	4.2579E-03
Year 10.00	5.5801E-07	1.6716E-04	3.9862E-03
Year 11.00			
Year 12.00			
Year 13.00			

You may change the size of the table fields by dragging boundaries of the field names and use the scroll-bar at the bottom of the table to view all results.

The list of variables for the **Accumulated Balance** differs from that of the **Balance** table:



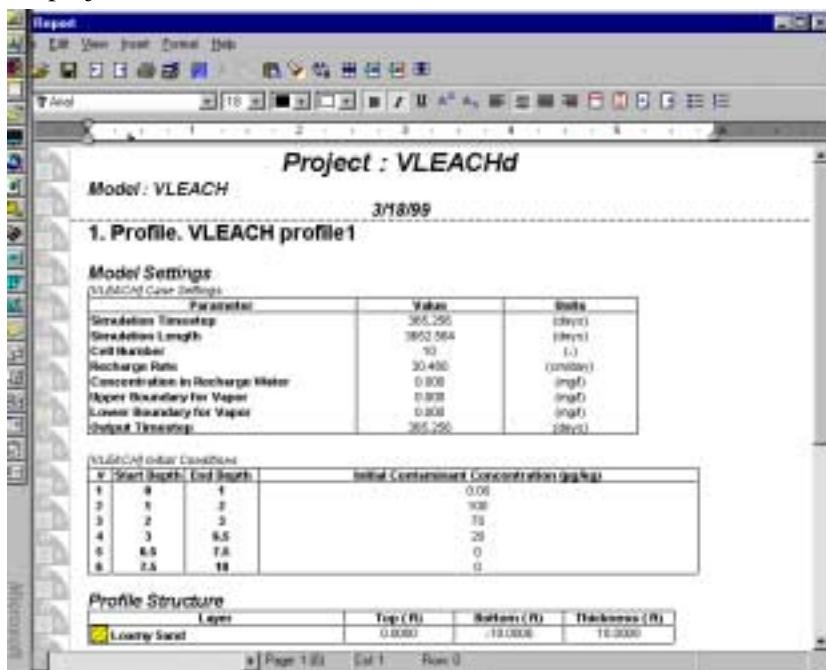
From this list, two variables; **Advection to ground water** and **Diffusion to ground water** will be of specific interest to hydrogeologists and environment protection specialists.

You may apply all viewing tools described above, for the **Balance** table and the **Accumulated Balance** table.

Preparing a Report

To present results of your VLEACH simulation to your clients you may use the UnSat Suite Report Generator.

[ To create a report and to add the project input data, click the following icon from the Operational Icons tool bar. The report will appear in a separate window. By default, the Report Generator lists all input data for your project:



In the **Report** window you may edit the report, input your own text and add any type of graphics or table outputs produced by UnSat Suite.

Note: The graphs and tables will be placed at the insertion point.

To add a graph or a table to the report:

- [1] In the **Report** window place the cursor to position where you want your graph or table to appear in the report
- [2] Create a graph or table using one of the methods described above.
- [3] <right click> in the Result View.
- [4]  **Insert To Report.** The graph or table will appear in the report.

A graph may appear smaller than the original. To get the graph to the desired size, click the graph in the **Report** window and stretch it until it reaches the desired size.

A table may be longer than the Report window allows. In this case the table will be automatically wrapped.

Add necessary graphs and tables into the report and write your comments. You may insert a header and footer in your report, apply different fonts and styles while working in the **Report** window. To utilize these and other options, make corresponding selections from the main menu. After you are done, you may print the report or save it.

Part 6:

The SESOIL Model

Introduction

SESOIL is a popular US EPA model which is capable to simultaneously model water transport, sediment transport and pollutant fate. SESOIL is widely used by consultants and state regulatory agencies as a screening tool to assess unsaturated zone contaminant fate and transport for regulatory requirements.

Allowed profile structure: four-layer homogeneous regarding flow and variable regarding transport profile with initially uneven distribution of contaminant.

Simulated processes:

Surface: weather boundary condition, surface runoff, soil erosion, pollutant washload

Subsurface: variable flow, as a function of the average moisture content, and advective transport of the contaminant through soil affected by sorption, volatilization, degradation, cation exchange, hydrolysis and metal complexation.

SESOIL is a 1-D compartment balance model. The code simulates leaching in a soil polygon. The contaminant may be present in the soil as initial condition and may be introduced or removed from any layer at any time during the simulation.

The total mass of contaminant within each cell is partitioned among three phases: liquid (dissolved in water), vapor, and sorbed to solid surfaces.

The SESOIL model proved to be a very efficient tool for screening assessment of potential groundwater contamination. The ability of the model to account for contaminant washload, volatilization and air diffusion of the volatile organic contaminants, sorption, volatilization, degradation, cation exchange, hydrolysis and metal complexation makes it a unique tool which only can be applied in many practical cases. This model has also a lot of advantages to be used for risk assessment studies which require multi-variant simulations.

UnSat Suite allows to quickly prepare, run and interpret SESOIL simulation by use of graphical tools. SESOIL is also equipped with a limited database of soil materials and agricultural and industrial contaminants.

15

Input Specification

This chapter describes how to setup a SESOIL project using specific parameter groups. To learn how to open a SESOIL project, please see Chapter 2.

Profiles in SESOIL

The SESOIL profile represents the unsaturated zone consisting of one material with homogeneous flow properties. In the hydrologic cycle, the whole column is treated as a single compartment extending from the soil surface to the groundwater table.

The profile is separated into 2, 3 or 4 **layers**, which may have different thicknesses. Layers are used to setup different transport parameters and pollutant loads within the profile. As designed by the code developers, the user has to specify flow and transport parameters for the uppermost layer only. As stated previously, the flow properties are the same for each layer of the model. Individual transport parameter settings for the lower layers are determined by parameter value ratios for each specific layer.

Each layer may be separated into 10 **sublayers** of equal thickness within a layer. Initial pollutant concentration may be set for each sublayer individually. In SESOIL, flow calculations between sublayers represents the highest level of detail. In addition, these sublayers are used to output simulation results.

In SESOIL, complex surficial and unsaturated zone processes are grouped into three cycles:

- Hydrologic cycle,
- Washload cycle,
- Pollutant fate cycle.

The hydrologic cycle is simulated first, followed by the washload cycle. The results from the hydrologic cycle and the washload cycle are then used to calculate the pollutant fate cycle. The hydrologic cycle is based on a statistical dynamic theory of a vertical soil water budget formulated by Eagleston (1978), which was adopted to account for variations in soil moisture. This hydrologic cycle controls the sediment and washload cycles. The pollutant cycle simulates contaminant transport and

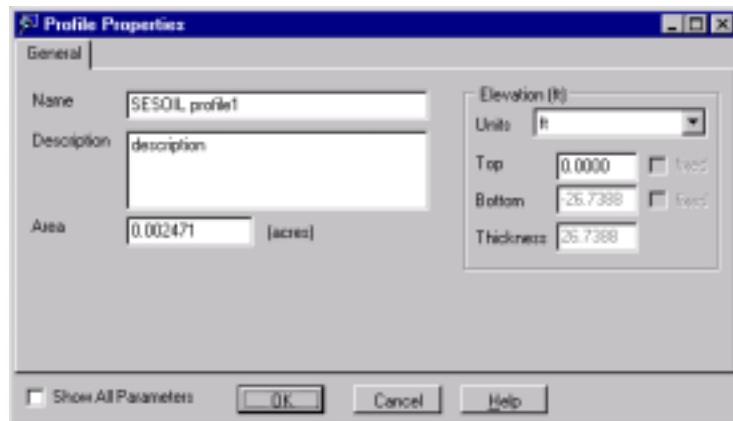
transformation in three soil phases: soil-air or gas phase, soil-moisture phase and soil-solid phase.

SESOIL requires time-variable boundary conditions and sources, and performs calculations using constant time intervals of 30.4 days (average month). The minimum simulation length for SESOIL is one year, which starts in October and ends in September.

Editing General Profile Properties

To edit the general profile properties, the **Profile Properties** dialogue window has to be opened.

☞ the profile **name** in the Project Tree View. The picture resembling the following will appear:



In this window you may assign the elevation of the profile top, change units for elevation, profile name or add a profile description.

In the **Area** text window you may input the area of site. The value of the area is used to calculate the profile balance.

Specifying the Case Settings

The **Case Settings** section contains parameters that affect the entire model profile. These settings are used to configure the SESOIL model. The Case Settings can be accessed from the Project Tree on the left hand side of the window.

To open the Case Settings parameter group:

- 1) the **Case Settings**,
- 2) OR <right click> the **Case Settings**.
- 3) Click **Edit**.

The following dialog box will appear:



The following parameters can be edited in the **Case Settings** parameter group:

Number of Layers

Total number of layers with different transport parameters and/or initial contaminant concentrations (minimum 2, maximum 4).

Note: Depending on number of layers you select, the number of rows in groups Contaminant Load Schedule and Initial Concentrations will be different.

Simulation Length

Total simulation length in years (to a maximum of 100).

Site Latitude

Site latitude.

Washload Simulation

Toggles the subroutine simulating soil erosion and contaminant washload On/Off.

To activate the washload function, click in the Value field and then select **Simulate Washload Transport**.

Note: If you activate the Washload function, two new parameter groups (Washload Settings and Washload Schedule) will appear in the Project Tree.

Spill Type

Specify contaminant loading as either instantaneous spill or continuous loading.

Month to load initial concentrations

Month during year 1 when initial contaminant concentrations are introduced. To add contaminant in subsequent years, see “Contaminant Application Schedule” on page 271.

Specifying Climate

SESOIL requires you to specify monthly climate parameter values for the entire simulation length. In the original SESOIL model, the user had to input the required data manually. However, WHI UnSat Suite provides a direct link between SESOIL and the WHI Global Weather Generator, allowing you to generate and import weather station data for nearly any location in the world. Please NOTE that climate data is “generated” based on the weather generator database, therefore the calculated data may vary slightly from the Weather Generator values.

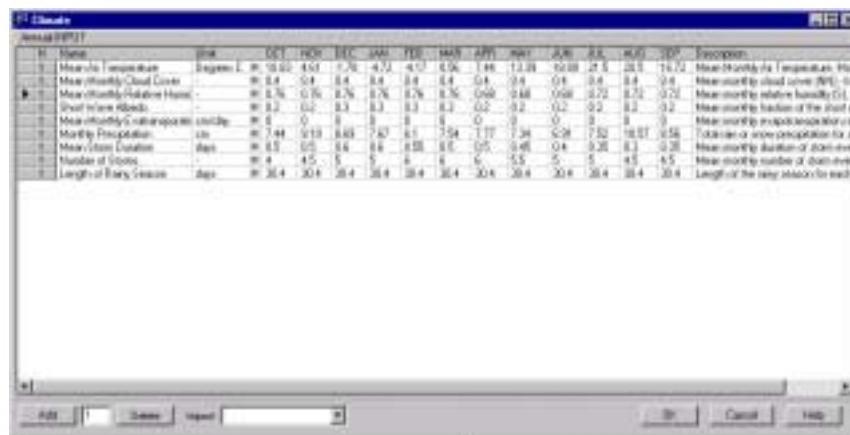
To set climate conditions:

<right click> on **Climate** in the Project Tree,

☞ **Edit**

OR

☞ ☞ on **Climate** in the Project Tree. The following dialog box will appear



This dialog box allows you to easily input and edit the appropriate number of annual climatic data sets.

Each annual set must contain the following parameters specified for each month:

Mean Air Temperature Mean monthly air temperature (Degrees C).

Mean Monthly Cloud Cover

Mean monthly fraction of cloud cover, ranging from 0.0 to 1.0.

Mean Monthly Relative Humidity

Mean monthly fraction of relative humidity, ranging from 0.0 to 1.0.

Short Wave Albedo

Mean monthly fraction of short wave albedo ranging from 0.0 to 1.0.

Mean Monthly Evapotranspiration Rate Mean monthly fraction of the evapotranspiration rate (cm/day).

Monthly Precipitation Total monthly precipitation (cm).

Mean Storm Duration Mean monthly duration of storm event (days).

Number of Storms Mean monthly number of storm events.

Length of Rainy Season Length of rainy season for each month of the year (days). For most regions this parameter should be set to 30.4, which means that rain may occur any day within the month.

The SESOIL model applies certain relationships between climate parameters. The parameters, **Mean Air Temperature**, **Mean Monthly Cloud Cover**, **Mean Monthly Relative Humidity** and **Short Wave Albedo**, are used only to simulate **Mean Monthly Evapotranspiration Rate**. This calculated rate is used by the model only if the **Mean Monthly Evapotranspiration Rate** is equal to zero.

Basing on the SESOIL code requirements, WHI has developed a technology that allows the user to easily set site specific climate data. The WHI UnSat Suite provides four options to input climate data in SESOIL:

- 1) input data manually,
- 2) input data from the Weather Generator database,
- 3) input weather data synthetically generated in SESOIL,
- 4) import synthetically generated weather data and evapotranspiration from HELP.

Each option requires a series of different steps that ultimately produces a set of climate data. Below you will find a description of the four options.

Inputting climate data manually

Using this option, the user must know all the required climate parameter values. To input the data, you must learn how to work with the **Annual INPUT** type of table within the SESOIL Interface.

Note: Beside CLIMATE, the Annual INPUT type of table is used for Washload Schedule and Contaminant Load Schedule parameter groups.

Working with the Annual INPUT table within the SESOIL Interface

By default, the **Annual INPUT** table opens with a one year set of climate parameters for Buffalo, NY. To input data manually, select the appropriate cell of the table and type the new value. You may change parameter units at any time for the appropriate parameter.

☞ **OK** after you have completed editing year 1. The **Annual INPUT** table will be saved. If you run the SESOIL model, the same annual set of climate data will be used for the number of years specified by parameter **Simulation Length** from the **Case Settings** parameter group.

If you want to enter data for more than one year, you may use the **Add** function which allows you to duplicate complete annual data sets.

To duplicate a single annual set:

- 1) ☞ in the left field beside any parameter in the desired annual data set. The entire annual set will be highlighted.
- 2) ☞ in the text-box beside the **Add** button and enter the number of copies.
- 3) ☞ **Add** button. The requested number of annual data sets will appear appended to the bottom of the table.

To duplicate several annual sets:

You may duplicate only sequential annual sets.

- 1) ☞ in the left field beside any parameter in the desired annual data set, click the left mouse button and move the pointer over the annual set you wish to multiply. The selected annual set will be highlighted.
- 2) ☞ in the text-box beside the **Add** button and enter the number of copies.
- 3) ☞ **Add** button. The requested number of annual set copies will appear appended to the bottom of the table.

Note: If the number of full annual data sets is less than the number of years specified in the Simulation Length, the SESOIL model will run the last annual set until the end of the simulation.

To delete one or several annual sets:

- 1) select one or more sequential annual sets as described above.
- 2) ☞ **Delete** button.

Input climate data from the Weather Generator database

This option allows you to create an annual data set with site specific average monthly values for Mean Air Temperature and Total Monthly Precipitation contained in the WHI Weather Generator database. This database contains data for more than 3000 locations around the Globe (see chapter 6 for information on the Weather Generator).

To use this option:

- 1) start the Weather Generator by selecting **Run/Weather Generator** from the main menu

OR

- ☞ the icon  in the main toolbar.

- 2) Run the Weather Generator for 1 year as described in Chapter 6.
- 3) Save generated weather data and close the Weather Generator.
- 4) Open the **Climate Annual Input** table as described in “Specifying Climate” on page 258.
- 5) ☞ the drop-down arrow in the **Import** drop-down list box and select **Use averages** from the list that appears. Monthly **Mean Air Temperature** and **Total Monthly Precipitation** for the specified weather station will appear in the appropriate rows of the first annual set.
Next you have to manually input monthly values for the remaining six parameters (except **Mean Monthly Evapotranspiration Rate**). Subsequently, you may multiply and edit annual data sets as described above (see “**Working with the Annual INPUT table within the SESOIL Interface**” on page 261).
☞ **OK** when completed.

Input weather data synthetically generated in SESOIL

This option allows you to synthetically generate site-specific statistically reliable data for the entire simulation length with the Weather Generator. This option allows you to assess values of **Mean Air Temperature**, **Total Monthly Precipitation** and **Number of Storms**. The remaining five parameters (except **Mean Monthly Evapotranspiration Rate**) have to be input manually.

To use this option:

- 1) start the Weather Generator by selecting **Run/Weather Generator** from the main menu

OR

- ☞ the icon  in the main toolbar.

2) Run the Weather Generator for **N+1** years, where **N** is the specified simulation length.

Note: N+1 years is required as the Weather Generator produces annual data sets that begin in January while the SESOIL year starts in October.

3) Save the generated weather data and close the Weather Generator.

4) Open the **Climate** Annual Input table.

5) \Rightarrow the drop-down arrow in the **Import** drop-down list box and select **Use simulated values** from the list that appears. Monthly **Mean Air Temperature**, **Total Monthly Precipitation** and **Number of Storms** for the specified weather station will appear in the appropriate rows of the **Climate** Annual Input table.

Next, you have to manually input the five remaining monthly parameter values (except **Mean Monthly Evapotranspiration Rate**).

\Rightarrow **OK** when completed.

Import synthetically generated weather data and evapotranspiration from the HELP model

This option requires you to use an additional WHI UnSat Suite model, HELP. However, this option will produce the most complete weather data for the entire length of the SESOIL simulation.

To use this option:

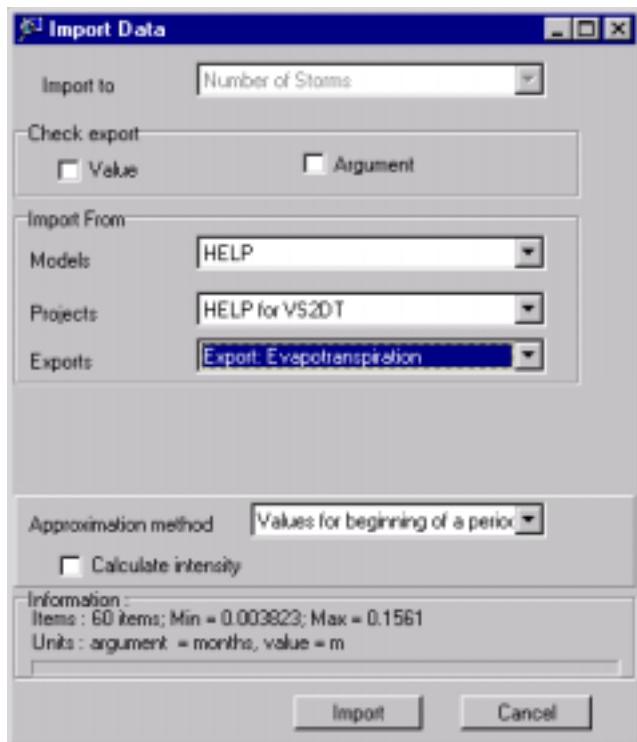
1) Create a Visual HELP project representing the unsaturated zone profile of your site (see Chapters 2, 5 and the Visual HELP Infiltration Lab exercise). Create a unit template with *months* units for time. Select this unit template for the model output.

2) Start the Weather Generator in the HELP model. Specify the Evapotranspiration parameter and run the Weather Generator for **N+1** years, where **N** is the specified SESOIL simulation length.

3) Run the HELP model for your site conditions. Open the output and export Evapotranspiration using the internal Unsat Suite format (see Chapter 17, Internal Data Transfer between WHI UnSat Suite Models).

4) Open your SESOIL project and the **Climate** Annual Input table.

- 5) Click the drop-down arrow in the **Import** drop-down list-box and select **Import from HELP** from the list that appears, as seen below:



- 6) If you have more than one Visual HELP project in the current project set, click the drop-down arrow to the right of the **Import from/Projects** drop-down list box and select the appropriate project.

The name of the exported Visual HELP variable (**Export: Evapotranspiration**) will appear in the **Import From/Exports** drop-down list box.

7) Click **Import**

Monthly Mean Air Temperature, Total Monthly Precipitation, Mean Monthly Evapotranspiration Rate and Number of Storms for the specified weather station will appear in the appropriate rows of the **Climate Annual Input** table.

The only parameter you have to input manually is **Mean Storm Duration**. However, this parameter does not change drastically from year to year, thus making data input relatively simple.

8) Click **OK** when complete.

Specifying Soil Erosion and Contaminant Washload

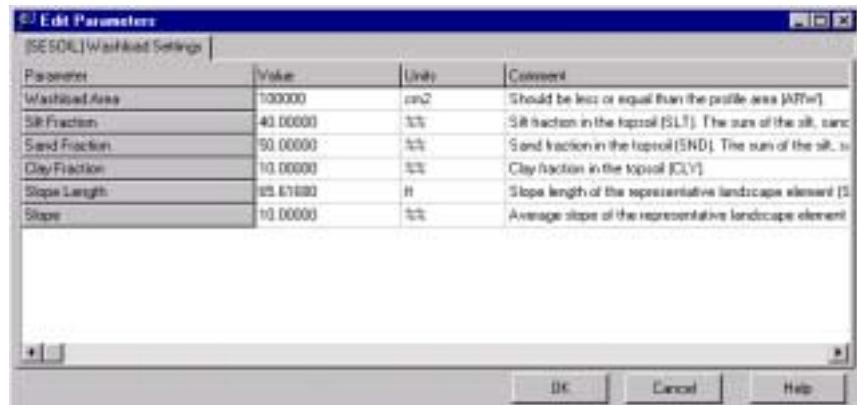
Two parameter groups are used to specify soil erosion and contaminant washload settings. The **Washload Settings** parameter group contains soil and landscape parameters that govern annual erosion and contaminant washload processes.

Specifying Washload Settings

To open the Washload Settings parameter group:

- 1) the **Washload Settings**,
- OR
- 2) <right click> the **Washload Settings** and Click **Edit**.

The following dialog box will appear:



The following parameters can be edited in the **Washload Settings** parameter group:

Washload Area	Washload simulation area; must be less than or equal to the profile area.
Silt Fraction	Silt fraction in the topsoil.
Sand Fraction	Sand fraction in the topsoil.
Clay Fraction	Clay fraction in the topsoil.

Note: The sum of the silt, sand and clay fractions must equal 1.

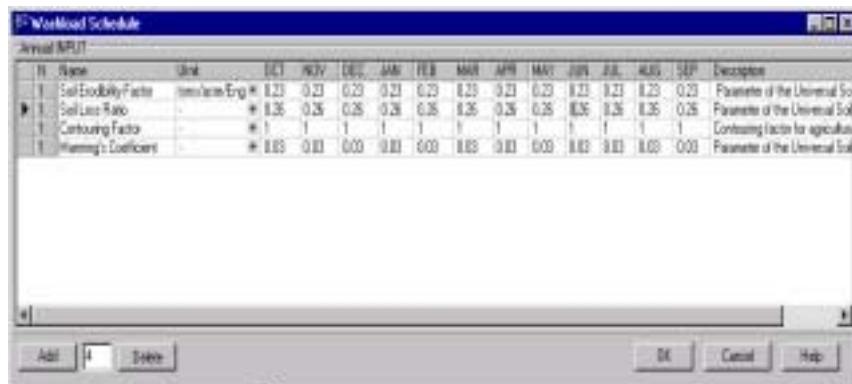
Slope Length	Slope length of the representative landscape element.
Slope	Average slope of the representative landscape element.

Specifying Washload Schedule

To open the Washload Schedule parameter group:

- 1) the Washload Schedule,
- OR
- 2)<right click> the Washload Schedule and Click Edit.

The following dialog box will appear:



The following parameters can be edited in the **Washload Schedule** parameter group:

Soil Erodibility Factor Universal Soil Loss Equation parameter, typically ranges from 0.03 to 0.69 (default set to 0.23).

Soil Loss Ratio Universal Soil Loss Equation parameter, typically ranges from 0.0001 (well managed soil) to 0.94 (tilled). Default set to 0.26.

Contouring Factor Contouring factor for agricultural land, typically ranges from 0.1 (extensive practice) to 1.0 (no supporting practice). Default set to 1.0.

Manning's Coefficient	Overland flow parameter in the Universal Soil Loss Equation, typically ranges from 0.01 to 0.4 (default set to 0.03).
------------------------------	---

Note: For more details on simulation of erosion and washload, see the original SESOIL manual on the installation CD. For information on how to edit annual data sets, see “Working with the Annual INPUT table within the SESOIL Interface” on page 261.

Specifying the Contaminant

This section describes how to setup initial conditions, specify contaminant parameters and define contaminant load schedule in the SESOIL model.

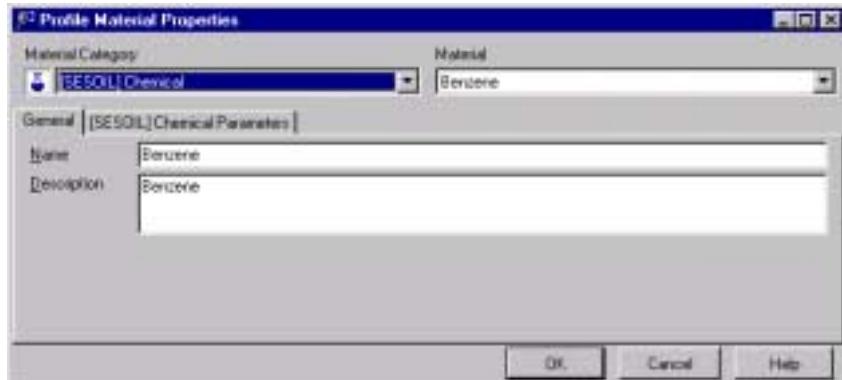
Defining the Contaminant

Each new SESOIL model profile starts with Benzene selected as the default contaminant. To define a different chemical as a contaminant for your model, open the **Profile Material Properties** dialogue box for the chemical.

To open Profile Material Properties dialog box:

- 1) the **name** of the chemical at the bottom of the Project Tree,
OR
- 2) <right click> the **name** of the chemical and click **Edit**.

The following dialog box will appear:



☞ the drop-down arrow of the **Material** list box. A list of 23 sample chemicals will appear:



Select a desired chemical, and edit the chemical parameters to match the conditions of your site.

In addition to using the available contaminants, you may add as many new chemicals to the database as required using the Material Designer (see Chapter 2).

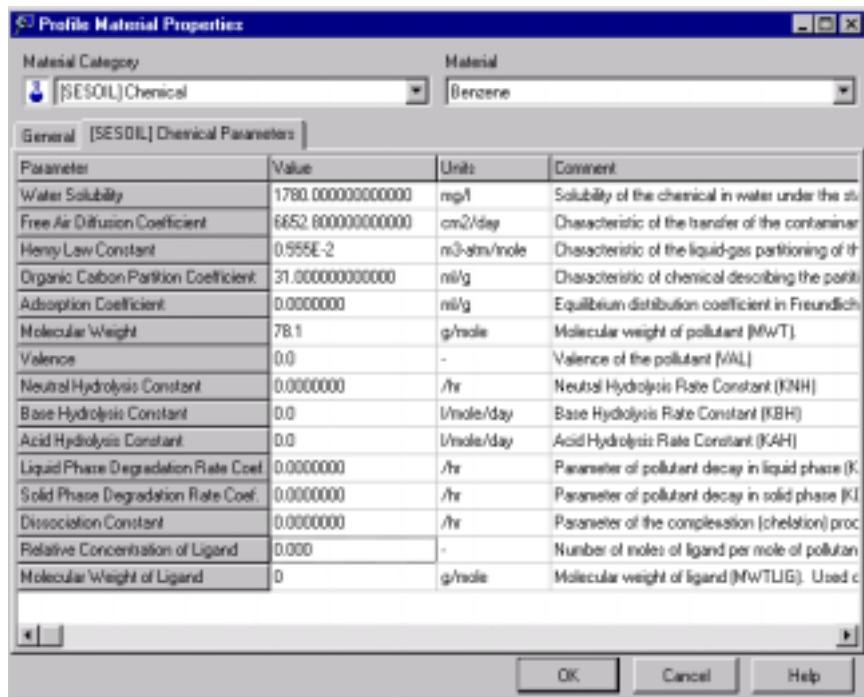
Editing Chemical Properties

You can access contaminant properties in two ways:

To access chemical properties:

- 1) ☞ ☞ the **name** of the chemical in the Project Tree,
☞ **Chemical Parameters** tab,
- OR
- 2) ☞ the **name** of the chemical and click **Edit**,
☞ **Chemical Parameters** tab.

The following dialog box will appear:



The following contaminant parameters can be modified in this dialogue:

Water Solubility	Solubility of the chemical in water under standard conditions.
Free Air Diffusion Coefficient	Characteristic of contaminant transfer due to Brownian motion in the air phase.
Henry's Law Constant	Chemical characteristic of the contaminant describing liquid-gas partitioning.
Organic Carbon Partition Coefficient	Chemical characteristic of the contaminant describing organic carbon partitioning.
Adsorption Coefficient	Equilibrium distribution coefficient in Freundlich equation (K). Equal to K _d if Freundlich exponent = 1.
Molecular Weight	Molecular weight of contaminant.
Valence	Valence of contaminant.
Neutral Hydrolysis Constant	Neutral hydrolysis rate constant.

Base Hydrolysis Constant	Base hydrolysis rate constant.
Acid Hydrolysis Constant	Acid hydrolysis rate constant.
Liquid Phase Degradation Rate Coef.	Contaminant decay parameter in liquid phase.
Solid Phase Degradation Rate Coef.	Contaminant decay parameter in solid phase.
Dissociation Constant	Complexation (chelation) process parameter. During this process, the pollutant combines with organic molecules (ligands) and forms stable complexes, which is effective only for heavy metals. Should be set to zero for other cases.
Relative Concentration of Ligand	Number of moles of ligand per mole of pollutant. Used only for processes of complexation, and should be set to zero for other cases.
Molecular Weight of Ligand	Molecular weight of ligand, which is used only for complexation processes. Should be set to zero for other cases.

Note: For more details on chemical properties, see the original SESOIL manual available on the installation CD.

Setting Initial Conditions

SESOIL allows you to specify the initial pollutant concentration in each sublayer within the profile.

To set initial contaminant concentration:

<right click> on **Initial Concentrations** in the Project Tree,

 **Edit**

OR

  on **Initial Concentrations** in the Project Tree.

j:\ Special Input groups for SESOIL model

- Table Initial Concentration

Concentration	SUBLAYER										
		[1]	[2]	[3]	[4]	[5]	[6]	[7]	[8]	[9]	[10]
Layer 1		0.000	0.000	0.000	0.000	0.000	0.000	0.000	0.000	0.000	0.000
Layer 2		0.000	0.000	0.000	0.000	0.000	0.000	0.000	0.000	0.000	0.000
Layer 3		0.000	0.000	0.000	0.000	0.000	0.000	0.000	0.000	0.000	0.000
Layer 4		0.000	0.000	0.000	0.000	0.000	0.000	0.000	0.000	0.000	0.000

OK Cancel Help

Enter the initial contaminant concentration in the appropriate cells, changing the units where necessary.

Note: The number of active sublayers is regulated by the Layer Parameters group settings.

☞ [OK] to save data and close the dialog box.

Contaminant Application Schedule

In SESOIL, you may input contaminant into the profile or simulate a contaminant sink from a specified layer at any time during the simulation.

To set the Contaminant Load Schedule:

☞ the **Contaminant Load Schedule** group in the Project Tree View,

OR

<right click> the **Contaminant Load Schedule** and click **Edit**.

The following SESOIL Annual Input group will appear:

Name	Unit	JAN	FEB	MAR	APR	MAY	JUN	JUL	AUG	SEP	OCT	NOV	DEC	TEF	Description
-year1 Pollutant Load	grd2	0	0	0	0	0	0	0	0	0	0	0	0	0	Monthly mass of pollutant load in the top sublayer.
-year1 Pollutant Transformation	grd2	0	0	0	0	0	0	0	0	0	0	0	0	0	Monthly mass of pollutant transformed in the top sublayer.
-year1 Pollutant Removal	grd2	0	0	0	0	0	0	0	0	0	0	0	0	0	Monthly mass of pollutant removed from the top sublayer.
-year1 Ligand Input	grd2	0	0	0	0	0	0	0	0	0	0	0	0	0	Monthly input of ligand to the top sublayer.
-year1 Volatilization Index		1	1	1	1	1	1	1	1	1	1	1	1	1	Volatilization index of the layer.
-year1 DOLF Index		0	0	0	0	0	0	0	0	0	0	0	0	0	Index of pollutant concentration.
-year1 Pore Ratio	grd2	0	0	0	0	0	0	0	0	0	0	0	0	0	Ratio of pollutant concentration.
-year1 Pollutant Load	grd2	0	0	0	0	0	0	0	0	0	0	0	0	0	Monthly pollutant load in the top sublayer.
-year1 Pollutant Transformation	grd2	0	0	0	0	0	0	0	0	0	0	0	0	0	Monthly mass of pollutant transformed in the top sublayer.
-year1 Pollutant Removal	grd2	0	0	0	0	0	0	0	0	0	0	0	0	0	Monthly mass of pollutant removed from the top sublayer.
-year1 Ligand Input	grd2	0	0	0	0	0	0	0	0	0	0	0	0	0	Monthly input of ligand to the top sublayer.
-year1 Volatilization Index		1	1	1	1	1	1	1	1	1	1	1	1	1	Volatilization index of the layer.
-year1 Pollutant Load	grd2	0	0	0	0	0	0	0	0	0	0	0	0	0	Monthly pollutant load in the top sublayer.
-year1 Pollutant Transformation	grd2	0	0	0	0	0	0	0	0	0	0	0	0	0	Monthly mass of pollutant transformed in the top sublayer.
-year1 Pollutant Removal	grd2	0	0	0	0	0	0	0	0	0	0	0	0	0	Monthly mass of pollutant removed from the top sublayer.
-year1 Ligand Input	grd2	0	0	0	0	0	0	0	0	0	0	0	0	0	Monthly input of ligand to the top sublayer.
-year1 Volatilization Index		1	1	1	1	1	1	1	1	1	1	1	1	1	Volatilization index of the layer.
-year1 Pollutant Load	grd2	0	0	0	0	0	0	0	0	0	0	0	0	0	Monthly pollutant load in the top sublayer.
-year1 Pollutant Transformation	grd2	0	0	0	0	0	0	0	0	0	0	0	0	0	Monthly mass of pollutant transformed in the top sublayer.
-year1 Pollutant Removal	grd2	0	0	0	0	0	0	0	0	0	0	0	0	0	Monthly mass of pollutant removed from the top sublayer.
-year1 Ligand Input	grd2	0	0	0	0	0	0	0	0	0	0	0	0	0	Monthly input of ligand to the top sublayer.
-year1 Volatilization Index		1	1	1	1	1	1	1	1	1	1	1	1	1	Volatilization index of the layer.

In this table, you may specify contaminant sources/sinks for any layer for each month of the simulation. By default, the input template for year 1 is opened.

The following contaminant sources/sinks may be specified:

Pollutant Load

Monthly pollutant load in the top sublayer (e.g. leaking storage tank, or pesticide application to soil). The Initial Concentration value entered previously refers to the contaminant concentration at the start of the simulation. If you have additional contaminants being added to the profile after time=0, the Pollutant Load variable can accommodate this. Otherwise, “year 2” initial concentration would be the output of “year 1”.

Pollutant Transformation

Monthly mass of pollutant transformed in the layer.

Pollutant Removal

Monthly mass of pollutant removed from the layer.

Ligand Input

Monthly input of ligand to the layer.

Volatilization Index

Volatilization index of the layer. VOLF=0.0 means no volatilization/diffusion from the layer, while VOLF=1.0 means full volatilization/diffusion from the layer.

Runoff Index Ratio of pollutant concentration in surface runoff to concentration in top sublayer.

Rain Ratio Ratio of pollutant concentration in rain to pollutant solubility in water.

Note: For information on how to edit annual data sets, see “Working with the Annual INPUT table within the SESOIL Interface” on page 261.

Modifying the Profile

Profile Properties

To view or edit the profile properties:

<right click> the profile picture, and click **Profile Properties**,

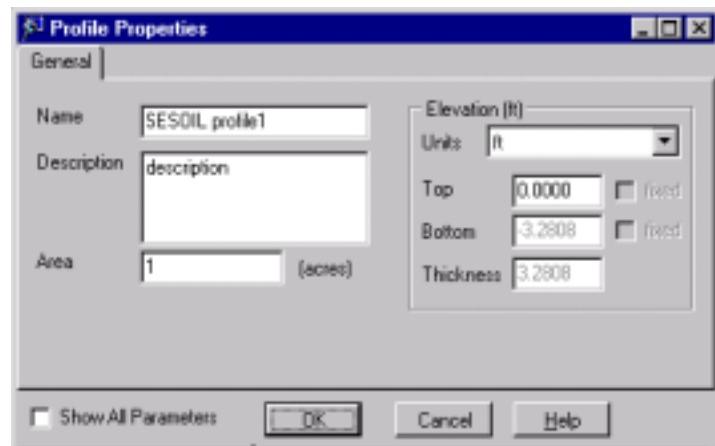
OR

“**Profile Properties**”

OR

<right click> the profile name and click **Profile Properties**.

The **Profile Properties** dialogue box will appear:



The following data is available in the **Profile Properties** dialogue box:

Name Type the profile name.

Description Type a profile description.

Elevation	Specify the top elevation of the profile.
Area	Specify the land area represented by this profile.

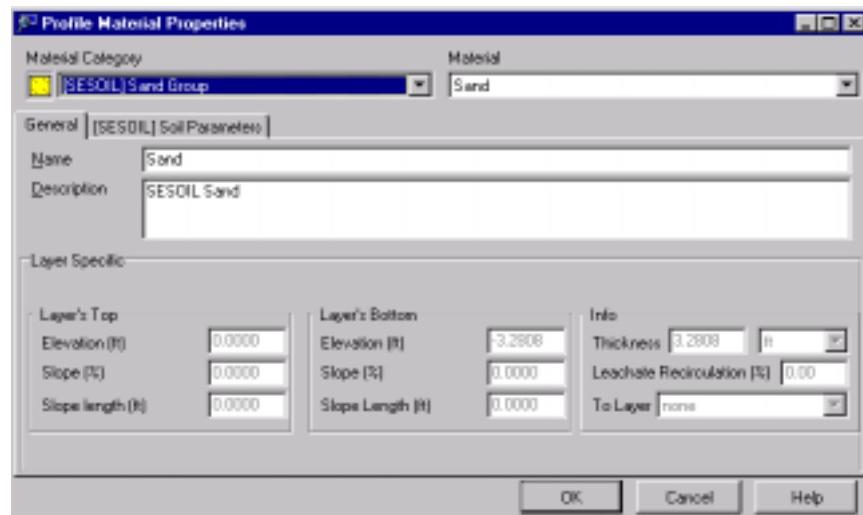
Setting the Profile Material

The SESOIL profile represents the unsaturated zone consisting of one homogeneous material extending from the soil surface to the groundwater table (i.e. the water table is the bottom of the profile).

To view the parameters of the profile material:

- 1) <right click> on the soil in the Profile View.
 - 2) Click **Layer/Properties**. The **Profile Material Properties** dialogue box for the soil will open,
- OR
- ☞ the soil name in the Project Tree View,
- OR
- <right click> the soil name and click **Properties**.

The following dialog box will appear:



This dialog box provides you with information on the profile material. To match the soil profile to your case, you may either substitute the material with one from the SESOIL database or edit the material parameters.

Substituting the Profile Material

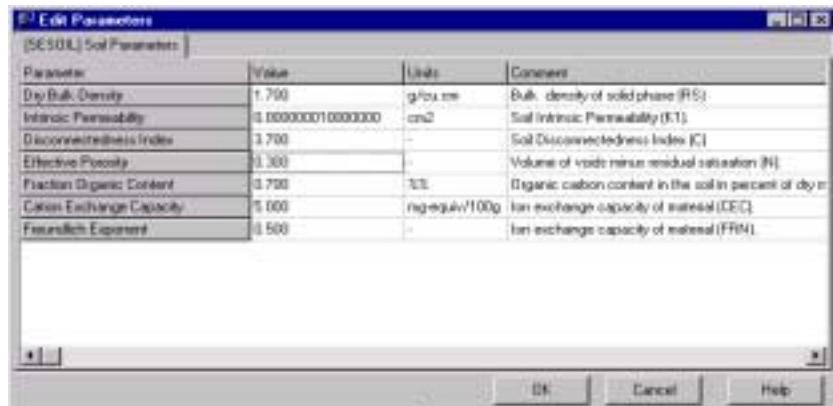
To substitute a material:

- 1) Select a new material category from the **Material Category** list. The available soils will appear in the **Material** drop-down list box.
- 2) Select a new soil (material) from the **Material** list.
- 3) Assign a unique name to the new layer and enter a descriptive comment.
- 4) Click the **Soil Parameters** tab and edit the soil parameter values if necessary.
- 5) Click **[OK]**.

Editing Soil Properties

Open the **Profile Material Properties** dialogue box using one of the methods described in the previous section.

☞ the **Soil Parameters** tab. The following dialog box will appear:



The following soil parameters can be modified:

Dry Bulk Density	Dry bulk soil density.
Intrinsic Permeability	Intrinsic soil permeability (cm^2), which can be converted to saturated hydraulic conductivity (cm/sec) by multiplying by $1.0 E+5$.
Disconnectedness Index	Soil Disconnectedness Index.
Effective Porosity	Volume of voids minus residual saturation.

Fraction Organic Content Soil organic carbon content in percent of dry matter weight.

Setting Layer Structure of the Profile

Although SESOIL is based on one homogeneous material, it may be separated into 2, 3 or 4 **layers** of different thicknesses. Layers are used to set different transport parameters and pollutant loads within the profile. As designed by the code developers, the user must specify flow and transport parameters for the uppermost layer only. Individual transport parameter settings for the lower layers are determined by parameter value ratios for the specific layer.

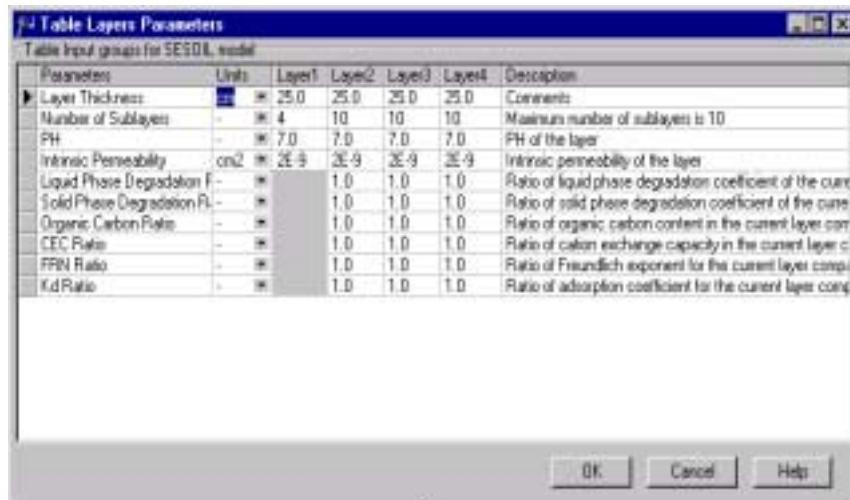
To set the layer structure of the SESOIL profile:

☞ the **Layer Parameters** group in the Project Tree View,

OR

<right click> the **Layer Parameters** and click **Edit**.

The following dialog box will appear:



Each column in this table specifies parameters for a specific layer. The number of columns in this table is specified by the **Number of Layers** in the **Case Settings** parameter group.

The following layer parameters can be modified:

Layer Thickness

Layer thickness.

Number of Sublayers	Number of sublayers used to input initial conditions and present simulation results. Maximum number is 10.
pH	pH of the layer.
Intrinsic Permeability	Intrinsic permeability of the layer.
<i>Note: The values of intrinsic permeability specified in this table are used only if this parameter is set to zero in the Soil Parameters table.</i>	
Liquid Phase Degradation Ratio	
	Ratio of the liquid phase degradation coefficient in the current layer compared to layer 1.
Solid Phase Degradation Ratio	
	Ratio of the solid phase degradation coefficient in the current layer compared to layer 1.
Organic Carbon Ratio	Ratio of organic carbon content in the current layer compared to layer 1.
CEC Ratio	Ratio of cation exchange capacity in the current layer compared to layer 1.
FRN Ratio	Ratio of Freundlich exponent for the current layer compared to layer 1.
Kd Ratio	Ratio of adsorption coefficient for the current layer compared to layer 1. Kd is set under Chemical Parameters as the Adsorption Coefficient, and is mostly used for inorganics.

Setting Groundwater Parameters

The SESOIL model allows you to set parameters of the aquifer. A modified Summer's model equation is used for computing the contaminant concentration in the saturated zone.

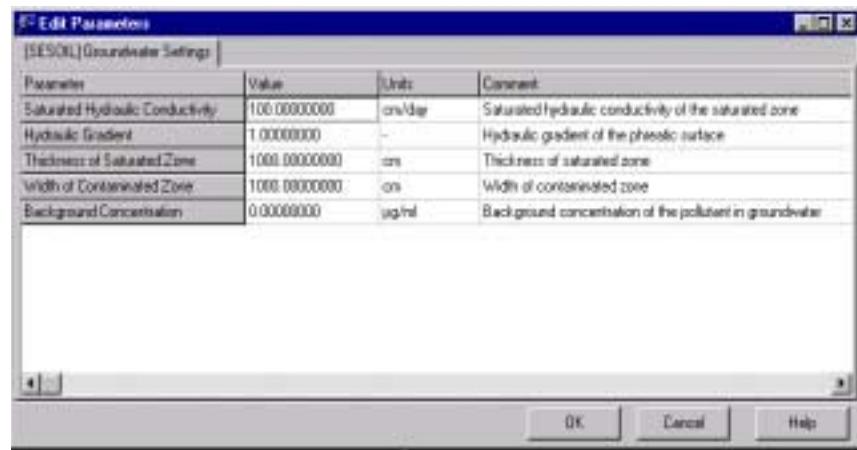
To set the Groundwater Parameters of the SESOIL profile:

☞ the **Groundwater Settings** group in the Project Tree View,

OR

<right click> the **Groundwater Settings** and click **Edit**.

The following dialog box will appear:



The following groundwater parameters may be edited in this group:

Saturated Hydraulic Conductivity

Saturated hydraulic conductivity of the saturated zone.

Hydraulic Gradient Hydraulic gradient of the phreatic surface.

Thickness of Saturated Zone

Thickness of saturated zone.

Width of Contaminated Zone

Width of contaminated zone.

Background Concentration

Background contaminant concentration in the groundwater.

16

Running the Model, Viewing Output and Reporting

Running the SESOIL Model



To run the model for a single profile, click the profile icon above the Profile View:

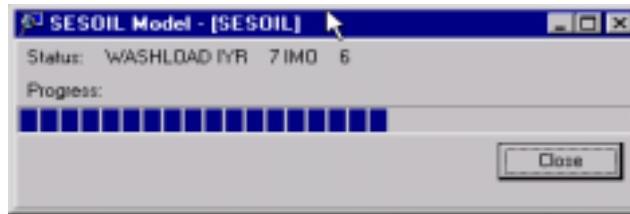


To run the model for multiple profiles or for one profile, if it is a single profile in the project, click the operational icon above the Project Tree View

OR

☞ **Run** in the main menu and then click **SESOIL**.

A progress bar will appear to indicate the computation progress, as shown below:



Viewing Original DOS SESOIL Output

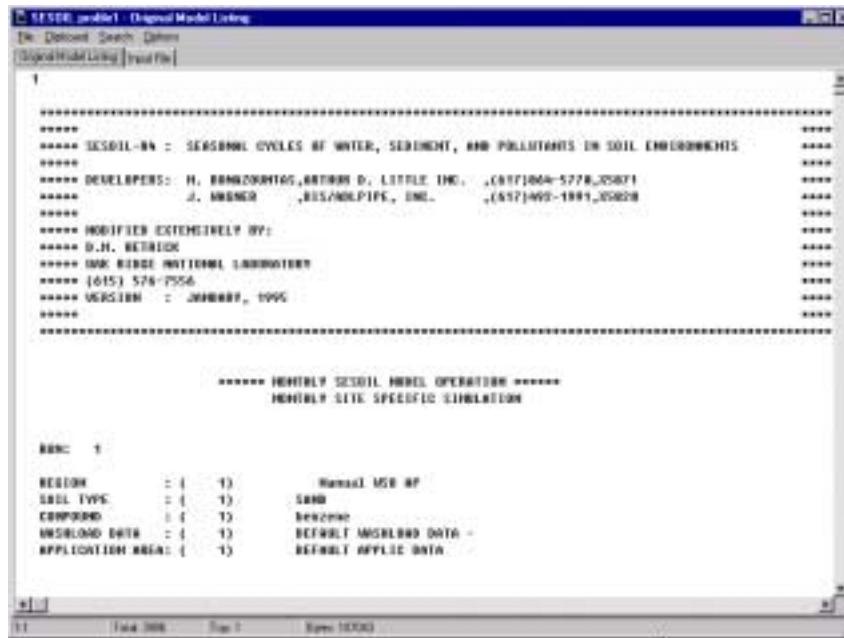
UnSat Suite allows you to view and print the original DOS **SESOIL** output.

To view and print original output:

☞ **Output** from the main menu.

☞ **Original Listing**

The **Original Model Listing** dialogue box will appear:



Scroll and view the original listing, find specific expressions, and print your results.

To view the SESOIL input file, click the **Input File** tab.

To print the file or the selected part of the file:

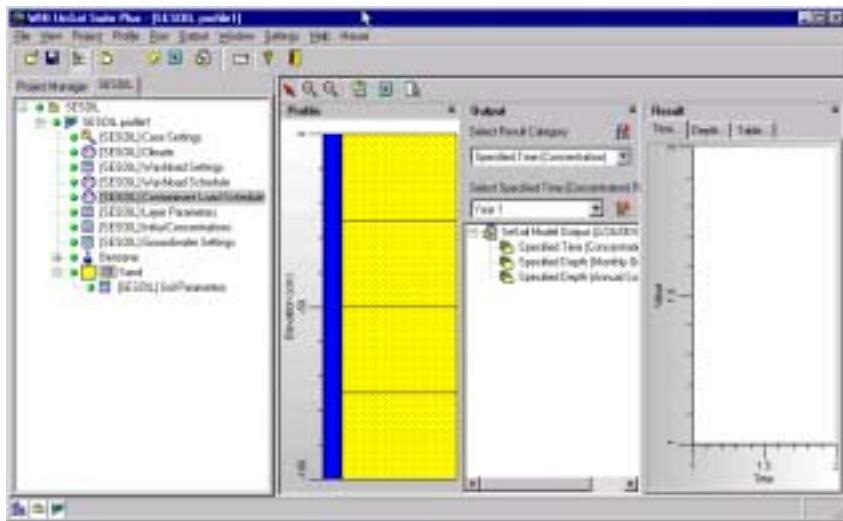
☞ **Print** from the **File** menu.

Specify the print properties, and click **[OK]** to print.

Click the **[X]** to close the window after you are finished.

Viewing the Output Graphs

After the model has successfully ran, the Output View and Result View windows will open and the UnSat Suite window will appear as shown below:



To enlarge the graph viewing area you may:



Click the icon to close the Project Tree View, or click the 'X' in the Profile View to close it.

The **Output** View window contains two drop-down windows:

1. **Select Result Category** (used to select general result category)
1. **Select Specified (Category Name) Result At....** (used to select specific result category)

To select general output category, click the arrow in the **Select Result Category** drop-down list box. The following list will appear:



Click the category you wish to view.

The first available result group for this category will appear in the listbox below. To view all available result groups, click the arrow in the **Select 'Name of Category' Result at...** drop-down listbox.

Specified Depth (Annual Summary)

This output category allows you viewing annual total balances of water and contaminant, as well as breakthrough curves at fixed depths (in specific sublayer) within the whole period of simulation.

As this output category allows you to make a general overview of simulation results, we recommend you always start your output examination with this option.

You may view the following sub-categories and variables as annual totals and averages:

Hydrologic Cycle Components *Components of the profile hydrologic balance*

Washload Cycle Components *Components of the profile washload cycle (if simulated)*

Pollutant Mass Input *Contaminant input into the profile presented for each layer*

Pollutant Mass Distribution *Contaminant balance presented for each layer and sub-layer*

Average Pollutant Concentration *Contaminant concentration in liquid, solid and gas phases of the soil presented for each layer and sub-layer*

Maximum Pollutant Depth *Contaminant downward propagation within the profile*

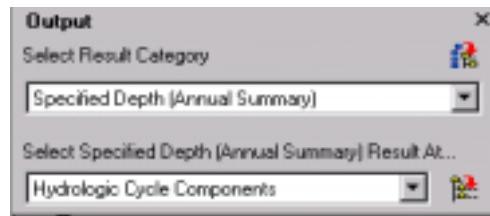
Average Concentration in Groundwater *Contaminant concentration in groundwater*

Next you will review some of the sub-categories.

Hydrologic Cycle Components

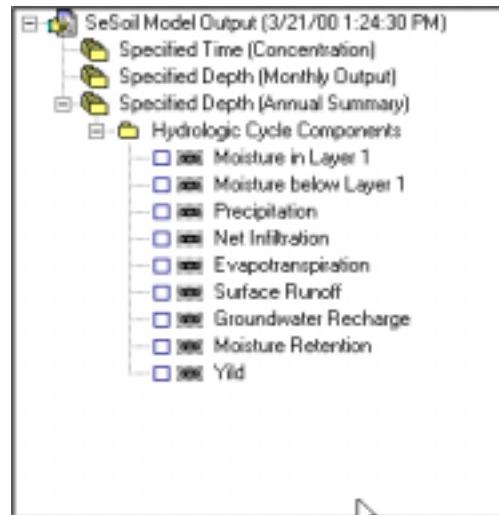
☞ Specified Depth (Annual Summary) in the Select Result Category drop-down list box.

The drop-down list box **Select Specified Depth (Annual Summary) Result at...** will show the first available sub-category **Hydrologic Cycle Components**:



To view results for this sub-category, click the icon to the right of the **Select Depth (Annual Summary) Result at...** box.

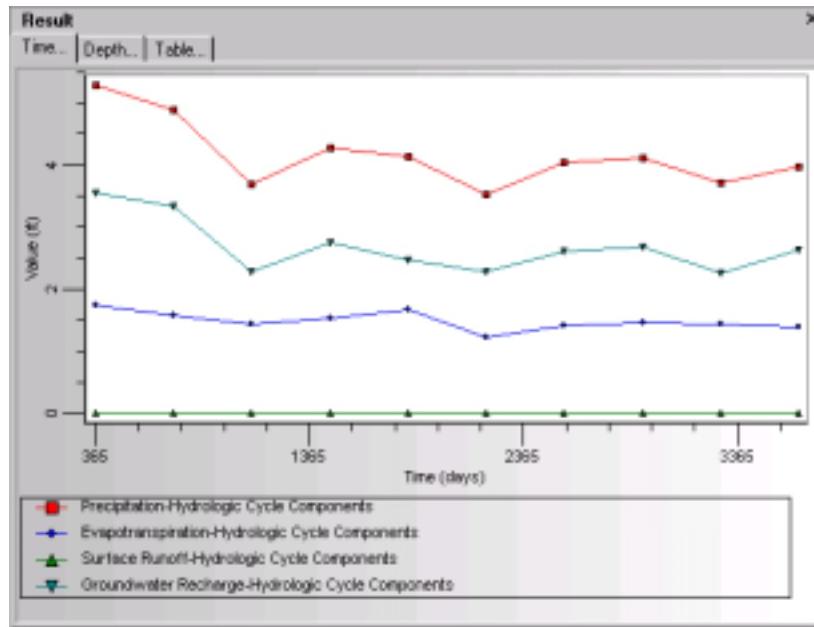
The list of available hydrologic variables will appear:



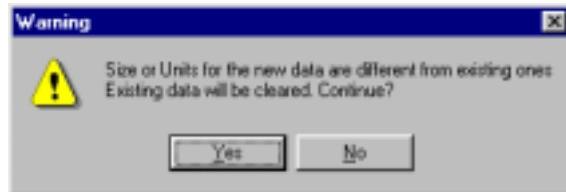
☞ the check-boxes beside the variables which you wish to view.

As far as you select check-boxes, graphs for specific variables will appear in the Result View window. Eventually, the picture resembling the one

below will appear if you select Precipitation, Evapotranspiration, Surface Runoff and Groundwater Recharge:

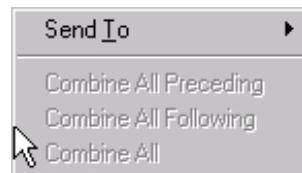


If you wish to view output for the variable which is measured in different units (e.g. Moisture in Layer 1), click the corresponding check box. The warning will be posted:



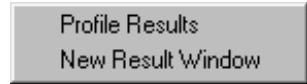
Select **Yes** if you want the previous graph to be replaced with the new one.

To view both variables which are measured in different units, you may place results for the second variable into the new Result Window. To do this, <right click> the name of the second variable (e.g. Moisture in Layer 1). The following menu will appear:



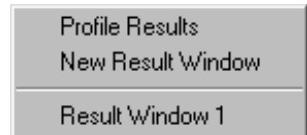
☞ **Send To**, an additional window will appear:

:



Choose **New Result Window**. Results for the second variable will appear in the **Result Window 1**. You may add graphs for the other times to the **Result Window 1** using the same method.

To see outputs for more than two variables, you may open additional Result Windows and place results there. To do this, <right click> the name of the additional variable. The following menu will appear:



Choose **New Result Window**. Results for the second variable will appear in the **Result Window 2**.

To clear **Result** window:

- ☞ **Output** (from the top menu bar)
- ☞ **Clear Display Results** (the Result View window will clear).

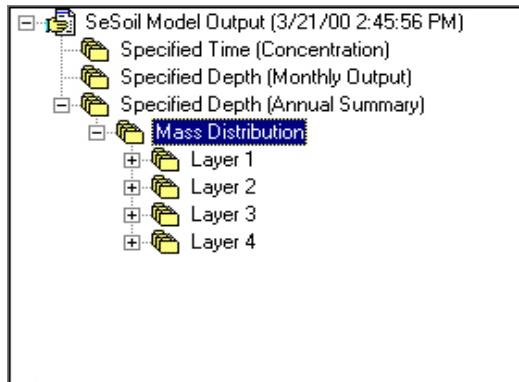
Contaminant Mass Distribution

- ☞ the arrow in the **Select Specified Depth (Annual Summary) Result at...** drop-down listbox. The list of available sub-categories will appear:



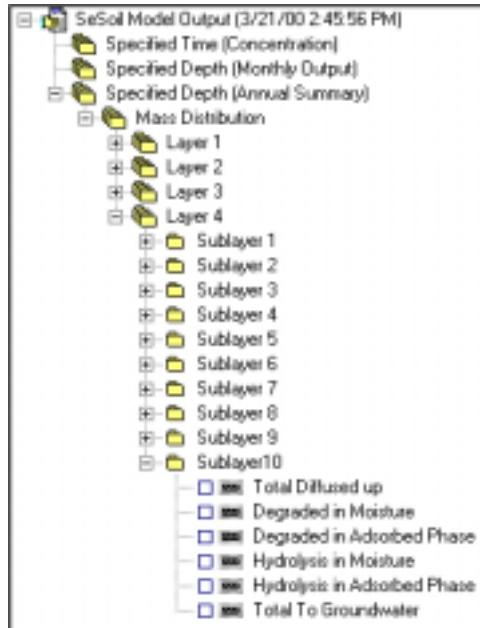
- ☞ **Mass Distribution.**

- ☞ the icon  to the right of the **Select Depth (Annual Summary) Result at...** box. The list of profile layers will appear in the window below:



Now you may study contaminant balance presented for each layer and sub-layer. Let, for example, study the contaminant distribution close to the groundwater level (in case you have all allowed layers and sublayers this refers to Layer 4, Sublayer 10) and volatilization to the atmosphere (this refers to Layer 1,Sublayer 1).

- ☞ the '+' sign to the left of **Layer 4**. Ten sublayers will appear in the tree.
- ☞ the '+' sign to the left of **Sublayer 10**. Available variables for this sublayer will appear in the tree:



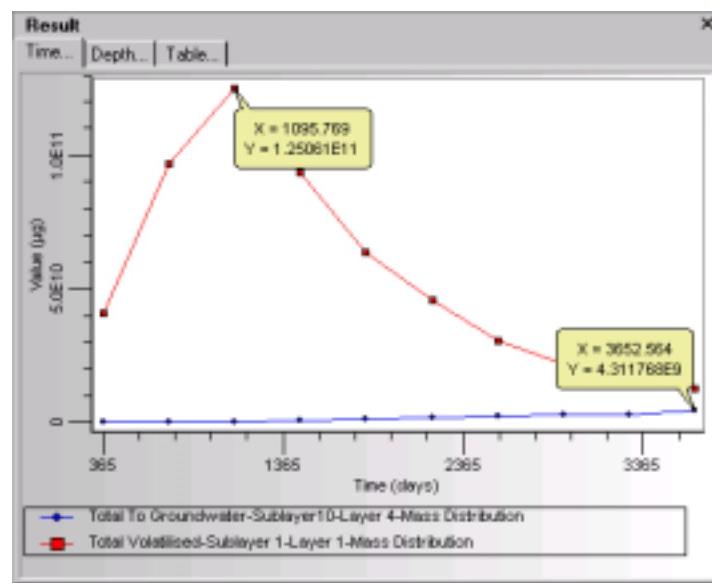
- ☞ the check box beside **Total To Groundwater**. The graph will appear in the **Result** view window.

- ☞ '+' sign to the left of **Layer 1**

- '+' sign to the left of **Sub-layer 1** (available variables will appear in the tree)
- the check-boxes beside **Total Volatilized**

The second graph will appear in the **Result** view window. To highlight the most important points at your graphs, move your mouse over the graph, place it on the peak values for each data set (a bubble sign will appear) and click. Values of the variable and the argument will appear in the bubble.

Your graph should appear resembling the figure below:



Pollutant Concentration

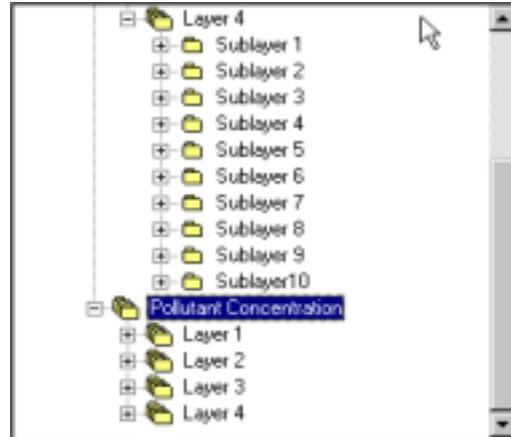
☞ the arrow in the **Select Specified Depth (Annual Summary) Result at...** drop-down listbox.

The list of available sub-categories will appear:



☞ Pollutant Concentration

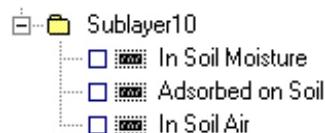
☞ the icon  to the right of the **Select Depth (Annual Summary) Result at...** box. The list of profile layers will appear in the Output Tree below the last viewed output. You may need to use the side slider in the Output Tree to reach this sub-category:



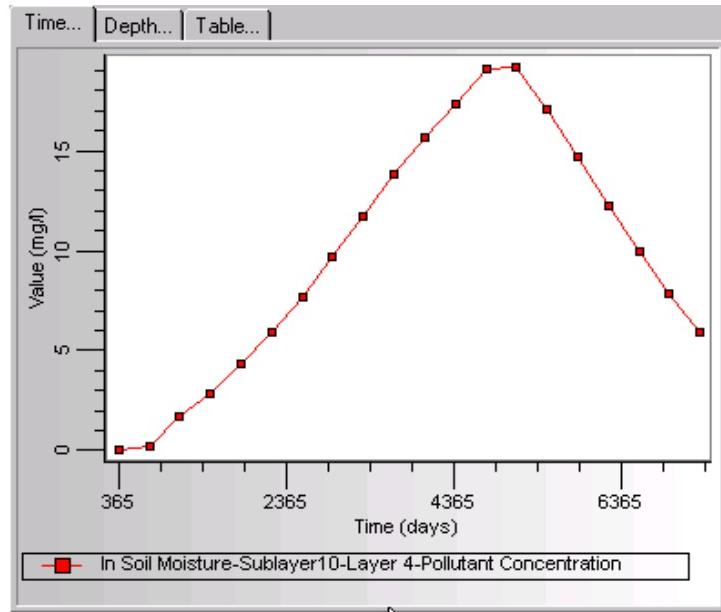
Now you may study contaminant concentration presented for each layer and sub-layer. Let, for example, study the contaminant concentration close to the groundwater level (in case you have all allowed layers and sublayers this refers to Layer 4, Sublayer 10).

☞ the '+' sign to the left of **Layer 4**. Sublayers of Layer 4 will appear in the tree.

☞ the '+' sign to the left of the bottom Sublayer. Available variables for this sublayer will appear in the tree:



☞ the check box beside **In Soil Moisture**. The graph for concentration, resembling the following, will appear in the **Output View** window:



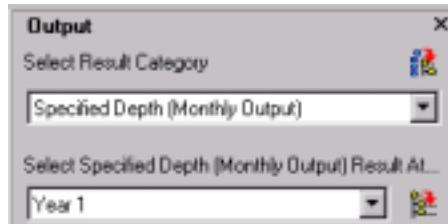
Viewing Specified Depth (Monthly Output)

This output category allows you viewing balances of water and contaminant, as well as breakthrough curves at fixed depth (in specific sublayer) within annual cycles of simulation.

All the methods of output viewing described above for the **Specified Depth (Annual Summary)** are applicable to the **Specified Depth (Monthly Output)** category. At the first level of output you have to select the **year** to view. Within the year output branch, the structure of the output for this general category are much the same as that for **Specified Depth (Annual Summary)**.

☞ **Specified Depth (Monthly Output)** in the **Select Result Category** drop-down list box. The first available annual set of results

Year 1 will show up in the drop-down list box **Select Specified Depth (Monthly Output) Result at...** below:



Let, for example, examine the pollutant concentration in the bottom of the profile (in case you have all allowed layers and sublayers this refers to Layer 4, Sublayer 10) for year 3.

☞ the arrow in the **Select 'Specified Depth (Monthly Output) Result at...** drop-down listbox.

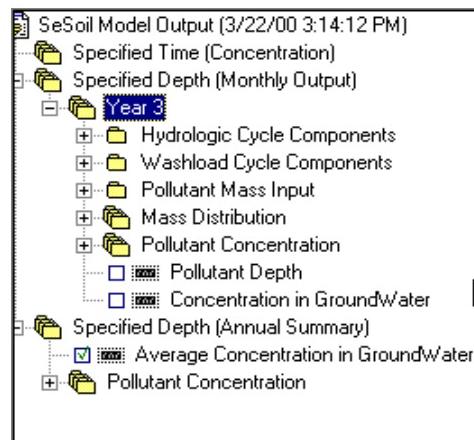
The list of available annual sets will appear:



☞ **Year 3** from the drop-down list.

☞ the icon  to the right of the **Select Specified Depth (Monthly Output) Result at...** drop-down list box to view available results for year 3.

The following structure will appear in the Output Tree:



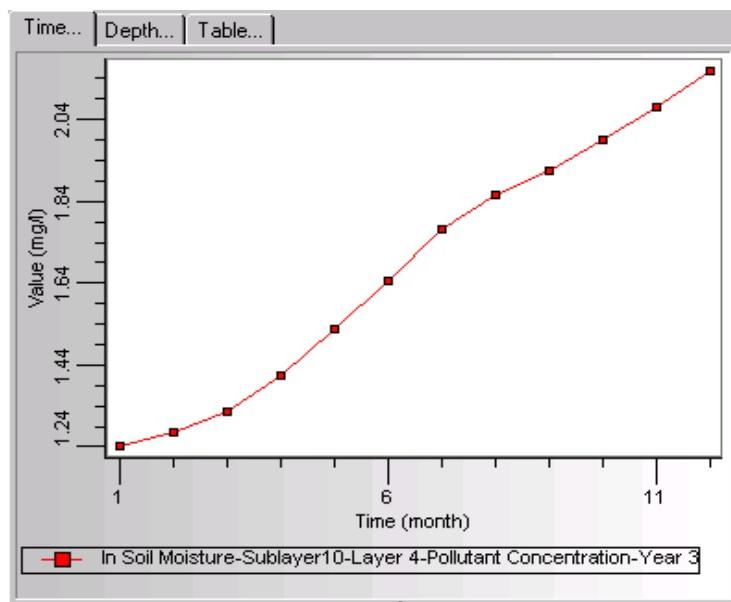
☞ the '+' sign to the left of **Pollutant Concentration** below **Year 3**.

The list of four profile layers will appear in the Output Tree.

☞ the '+' sign to the left of **Layer 4**. Ten sublayers will appear in the tree.

☞ the '+' sign to the left of **Sublayer 10**. Available variables for this sublayer will appear in the tree:

☞ the check box beside **In Soil Moisture**. The graph for concentration at Sublayer 10, Layer 4, the point closest to the groundwater surface, will appear in the **Output View** window:



Specified Time (Concentration)

This output category allows you viewing vertical profiles of contaminant concentration in liquid, gas and solid phases at any time during the simulation period.

☞ **Specified Time (Concentration)** in the **Select Result Category** drop-down list box.

☞ the arrow in the **Select 'Specified Time (Concentration)' Result at...** drop-down listbox.

The list of output times will appear:

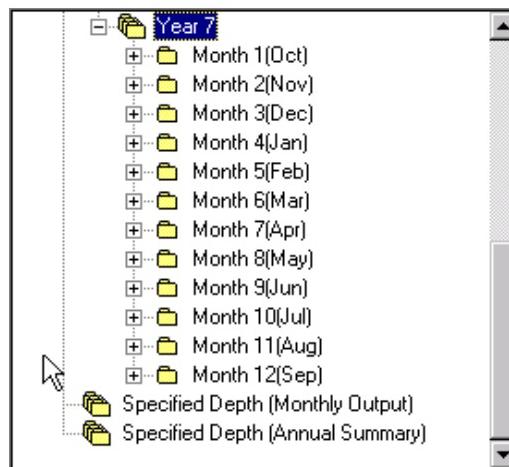


Use the slider to reach the time of your interest and click it. The selected time will appear in the drop-down box:

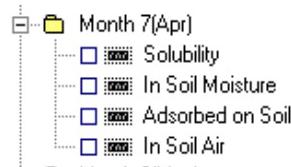


To view results for this specific time, click the icon  to the right of the **Select Specified Time Result at...** box.

The list of results for specified year will open in the Result Tree:



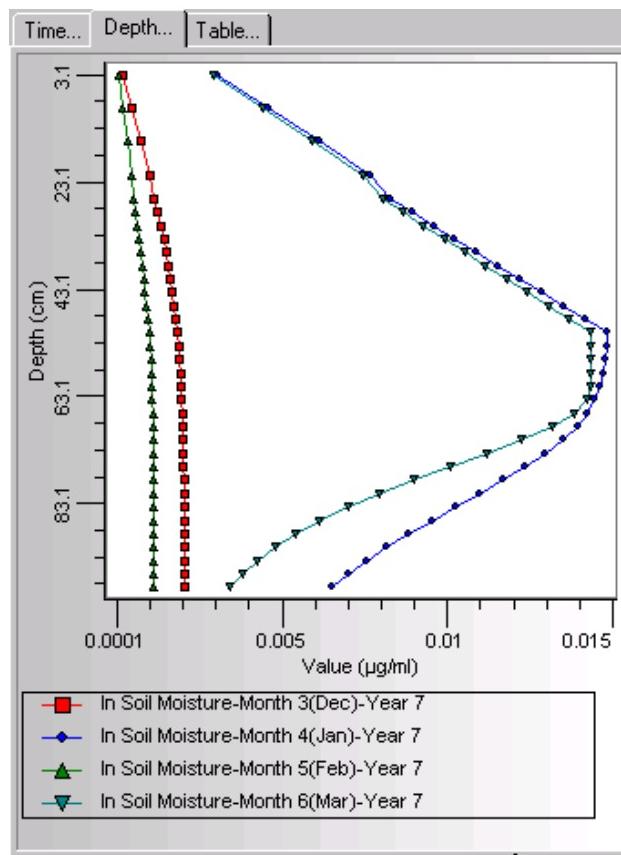
Click the add sign beside the month you wish to view results for:



Click the check box beside the type of variable you wish to view. The graph of the variable will appear in the Result View window.

To add the graph for another time to the same window, select a new time from the **Select Specified Time (Concentration) Result at...** box and check the same variables (a warning will display if you choose different

variables). The Result View will show profile distribution of the variable for different times:



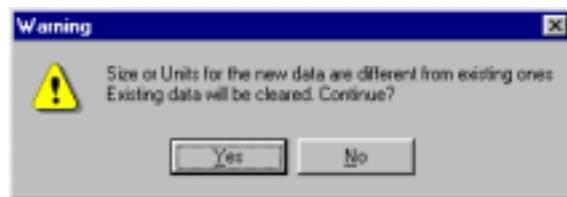
To erase an output for a specific time from the Result View window, unselect the corresponding check box in the Result Tree.

To clear the Result View window:

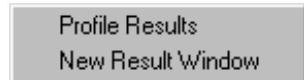
☞ **Output** from the main menu.

☞ **Clear Display Results**

If you wish to view output for other type of variable (e.g. Adsorbed on Soil), click the corresponding check box. The warning will be posted if the new and previous variables are measured in different units:

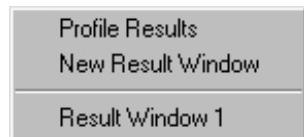


To view both variables which are measured in different units, you may place results for the second variable into the new Result Window. To do this, <right click> the name of the second variable. The following menu will appear:



Choose **New Result Window**. Results for the second variable will appear in the **Result Window 1**. You may add graphs for the other times to the **Result Window 1** using the same method.

To see outputs for more than two variables, you may open additional Result Windows and place results there. To do this, <right click> the name of the additional variable. The following menu will appear:



Choose **New Result Window**. Results for the second variable will appear in the **Result Window 2**.

Preparing a Report

To present results of your SESOIL simulation to your clients you may use the UnSat Suite Report Generator.



To create a report and to add the project input data, click the icon from the Operational Icons tool bar. The report will appear in a separate window.

By default, the Report Generator lists all input data for your project:

Project : SESOIL

Model : SESOIL
An LC-CPA model for step-wise simulations of chemical transport and transformations in soil.

Author : Your title Your name

Client : Title Key contact person /24/00

1. Profile. SESOIL profile1

Model Settings
(SESOIL) Case Settings

Parameter	Value	Units
Number of Layers	11	(-)
Simulation Length	3652.6	(days)
Site Latitude	35.0	(-)
Wasteload Simulation	Simulate wasteload transport	(-)
Spill Type	continuous	(-)
Month to load initial concentrations	1	(-)

[SESOIL] Climate 1-year

Var	Parameter	Unit	OCT	NOV	DEC	JAN	FEB	MAR
1	Mean Air Temperature	Degrees C	10.8300	4.8100	-1.7600	-4.7200	-4.1700	0.5600
1	Mean Monthly Cloud Cover	-	0.4000	0.4000	0.4000	0.4000	0.4000	0.4000
1	Mean Monthly Relative Humidity	-	0.7680	0.7800	0.7600	0.7600	0.7600	0.7600
1	Short Wave Albedo	-	0.2000	0.2000	0.3000	0.3000	0.3000	0.3000
1	Mean Monthly Extratranspiration Rate	mm/day	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000
1	Monthly Precipitation	mm	7.4400	9.1900	8.6900	7.6700	6.1000	7.5400
1	Mean Storm Duration	days	0.5000	0.5000	0.6000	0.6000	0.5000	0.5000
1	Number of Storms	-	4.8000	4.5000	5.0000	5.0000	6.0000	6.0000
1	Length of Rainy Season	days	20.4200	26.4000	20.4000	20.4200	20.4000	20.4000

In the **Report** window you may edit the report, input your own text and add any type of graphic output produced by UnSat Suite.

Note: The graphs will be placed at the insertion point.

To add a graph to the report:

- [1] In the **Report** window place the cursor to position where you want your graph or table to appear in the report
- [2] Create a graph using one of the methods described above.
- [3] <right click> in the Result View.
- [4] **Insert To Report.** The graph or table will appear in the report.

A graph may appear smaller than the original. To get the graph to the desired size, click the graph in the **Report** window and stretch it until it reaches the desired size.

Add necessary graphs into the report and write your comments. You may insert a header and footer in your report, apply different fonts and styles while working in the **Report** window. To utilize these and other options,

make corresponding selections from the main menu. After you are done, you may print the report or save it.

Part 7: Export and Internal Transfer of Simulation Results

17

Export , Internal Transfer, and Import of Simulation Results

This section describes tools that enable internal transfer of data between models within WHI UnSat Suite, as well as the export of model data for use with other applications.

Internal data transfer between WHI UnSat Suite models

WHI UnSat Suite combines different models that illustrate unsaturated zone processes from different viewpoints. The advantage of allowing different models to share the same interface is that, as a whole, the modeling package becomes more effective. Data produced by one model, and subsequently used as input into other model, can substantially improve the quality of the ultimate model output.

For example, VS2DT utilizes the constant or step-wise flux or head condition at the upper boundary. Commonly, it is hard for a practician to schematize existing information about soil boundary events in this form and prepare detailed input information describing transient processes. HELP, on the other hand, has a nicely developed routine for estimating surface and subsurface water flows, including infiltration and groundwater recharge. Import of the infiltration/evapotranspiration values determined by HELP, and their use as the upper flow boundary condition in VS2DT, strongly improves the reliability of results obtained with VS2DT.

Export from WHI UnSat Suite

Results obtained from one or more WHI UnSat Suite models can be exported from the Suite and used by other applications. For example, results exported from HELP, SESOIL and VS2DT models can be used as recharge rates and surficial pollutant loads for Visual MODFLOW model. However, some users might wish to export WHI UnSat Suite simulation results and import them into post-processing spreadsheet programs (e.g. MS EXCEL). All of the functions described above are now available with WHI UnSat Suite.

Internal Data Transfer between WHI UnSat Suite Models

HELP and VS2DT models have certain advanced features that increase accuracy and reliability of unsaturated zone simulation results when

combined. WHI has developed a technology that combines the advantages of each model, and allows them to work together in the most efficient way.

During the first stage of this technology, the upper part of the unsaturated zone (specific top layer or the root zone) is simulated with the HELP model and Weather Generator. This simulation provides high accuracy for assessment of water flux values at the bottom of the upper layer.

During the second stage, the VS2DT model is used to simulate unsaturated flow and transport of a specific chemical in the major (lower) part of the soil profile using results from the HELP simulation as an upper boundary condition.

To illustrate the application of this technology, we will examine a practical case to which WHI UnSat Suite was previously applied.

The case conditions are as follows:

Some innovative companies have developed processes for recycling powerplant bottom ash, the by-product of coal combustion. The processed ash product is used as construction fill to a depth of 2 feet at the site near Buffalo, NY. The ash contains chlorine, which can potentially leach from the ash and contaminate groundwater. During the first stage, HELP was used to assess the monthly values of water percolating through the 2 feet ash layer for a monthly intervals. The compacted aggregated ash has the following parameters: the saturated hydraulic conductivity 0.8 m/day, the total porosity 0.57 and field capacity 0.29. Initial conditions for soil profiles were near steady-state provided by the standard HELP procedure. The length of the simulation is ten years.

Note: *The length of Visual HELP simulation should be equal to the length of VS2DT forecast.*

During the second stage, the VS2DT model is used to simulate unsaturated transport of chlorine in the zone below the ash layer. The natural profile of the unsaturated zone below the ash layer is formed from a 10 m thick sandy loam layer. Standard VS2DT database flow and transport parameters are used for the sandy loam layer. The VS2DT application uses monthly averages of percolation through the ash layer assessed with Visual HELP as the flow upper boundary.

To conduct an internal transfer of data within the WHI UnSat Suite, the following steps need to be executed:

- i) Set output units in Visual HELP equal to the desired units of the model-recipient (VS2DT).
- ii) Run the Visual HELP model.

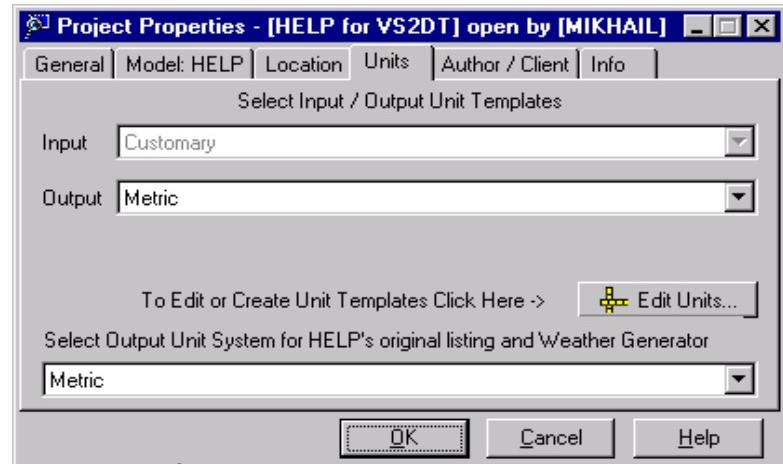
- iii) Open the Visual HELP output with the desired time intervals.
- iv) Export data from Visual HELP.
- v) Import data into the VS2DT model.

Below, you will find the description of additional functions required to transfer data between models. To view general instructions on how to setup the HELP project, see Chapter 2. To view general instructions on how to run the HELP project, see Chapter 7.

Setting output units in Visual HELP equal to the desired units of the model-recipient

Working with the WHI UnSat Suite you may select different units for the HELP and VS2DT models. The HELP model produces recharge values (percolation through the top layer) accumulated over a selected time interval (day, month or year) in units of length while VS2DT requires input in units of flux (m/day). In addition, the HELP model produces recharge values accumulated over a selected time interval (day, month or year) while VS2DT allows setting the time intervals for the flow upper boundary in days only. As such, you must be careful to check the consistency of Visual HELP and VS2DT units.

The recommended units for flow through the upper boundary (recharge rate) in VS2DT are ‘m/day’. To set these units for export from the HELP model, select the **Metric** unit template for HELP output when setting the project properties:



The metric Unit template has unit ‘m’ for length and ‘days’ for time.

Opening Visual HELP output with desired time intervals

After you create weather files with the Weather Generator and run the HELP model for the top ash layer, you have to open the data for percolation through the bottom of the ash layer with monthly time intervals.

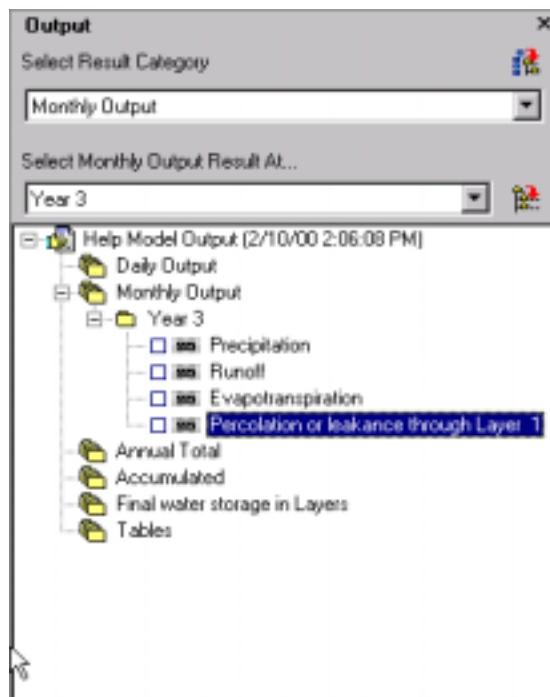
☞ the drop-down arrow in the **Select Result category** drop-down list box and then

☞ **Monthly Output**

☞ **Year 3** in the **Select Monthly Output Result at...**drop-down list box.

☞ the icon to the right of the **Select Monthly Output Result at...** box.

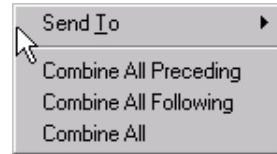
The list of available monthly variables for year 3 will open in the Result Tree:



Now select the check-box beside the **Percolation or leakage through Layer 1** in the Output Tree, as it is used in the WHI UnSat Suite to select variables. You will produce a graph of percolation through the bottom of the ash layer for year 3 in the **Result** window.

However, the task is to export recharge data for all 10 years of simulation. To accomplish this,

click the right mouse button at the name of the variable **Percolation or leakance through Layer 1**. The following dialog box will appear:



This dialog box allows you to select the data aggregation method.

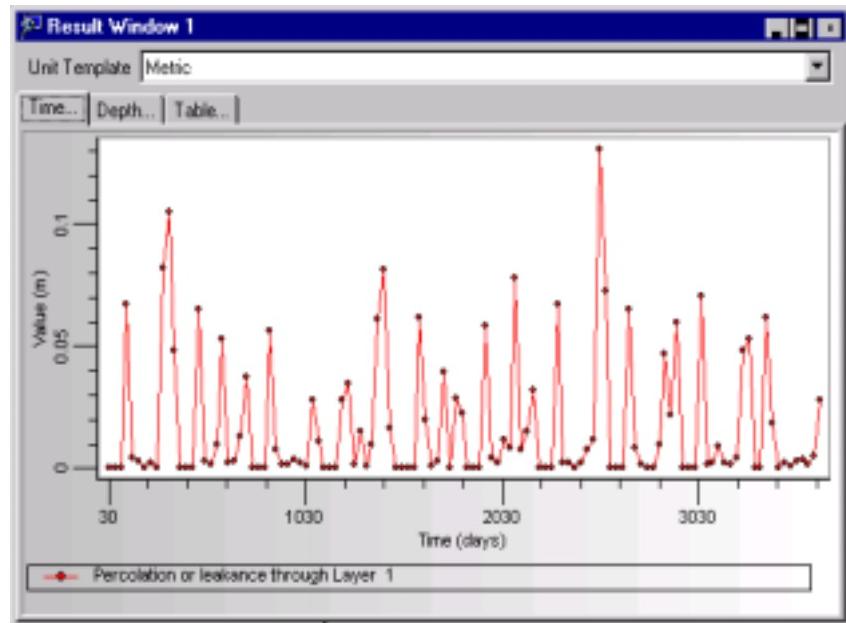
If you select **Combine All Preceding**, the data for all preceding years and the current year will be presented.

If you select **Combine All Following**, the data for the current year and all following years will be presented.

If you select **Combine All**, the data for the whole simulation period will be presented.

Combine All

The following graph will appear in the **Result Window 1**:



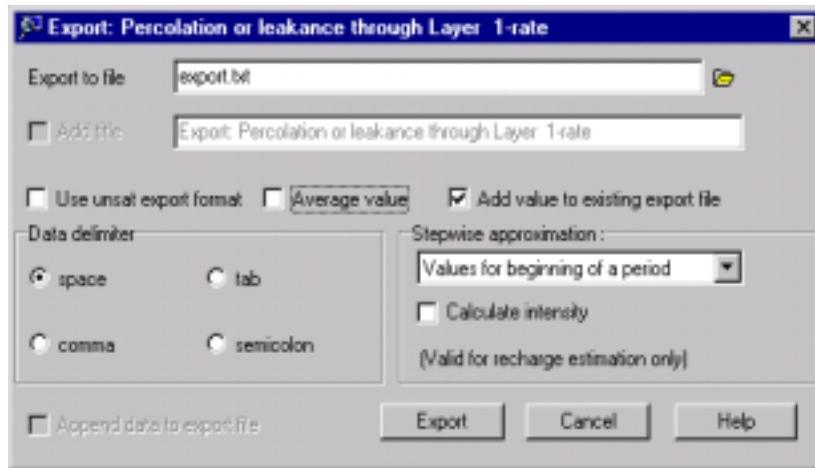
This graph shows monthly variation of percolation through the bottom of the ash layer over the 10-year period.

Exporting data from Visual HELP

To export data presented on the graph, place the mouse pointer at the graph line (the bubble sign will appear) and click right mouse button. A button named **Export..** will appear.

☞ **Export..**

The **Export: Percolation or leakance through Layer 1** dialog box will appear:



3) ☞ **Use unsat export format** check box.

4) ☞ the **Export** button.

Importing data into the VS2DT model.

To learn how to create a VS2DT project, see Chapters 2 and 10. After you create a VS2DT project representing the portion of the unsaturated profile below the ash layer, you may import data simulated by Visual HELP for flow through the ash layer.

In the VS2DT project tree:

☞ ☞ the **Flow Upper Boundary** parameter group. The following dialog box will appear:



☞ ☞ in the **Boundary type** field and select **Flux** from the list that appears.

☞ the **Import** button at the bottom of the dialog box. The following dialog box will appear:

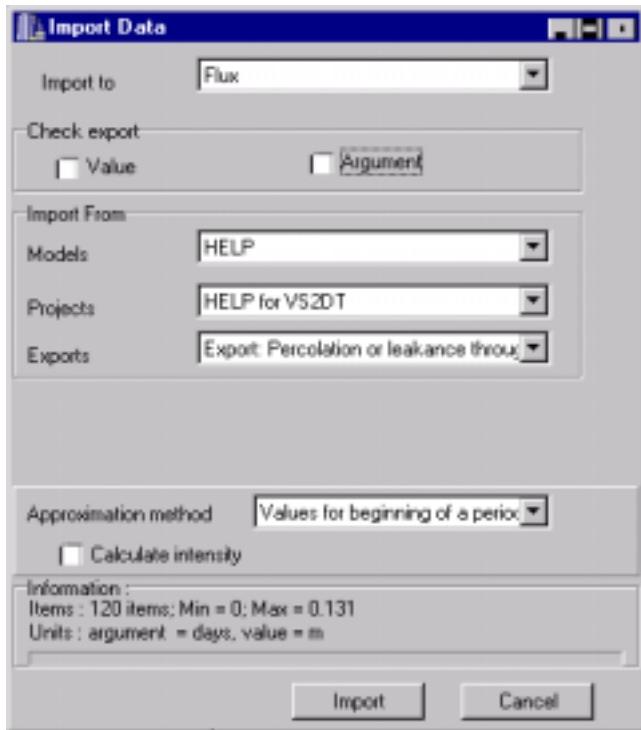


☞ the drop-down arrow to the right of the **Import to** drop-down list box.

☞ **Flux** from the list that appears.

If you have more than one Visual HELP project in the current project set,
☞ the drop-down arrow to the right of the **Import from/Projects** drop-down list box and select the appropriate project.

Unselect the **Check export/Value** and **Check export/Argument** boxes. The name of the exported Visual HELP variable will appear in the **Import From/Exports** drop-down list box.:

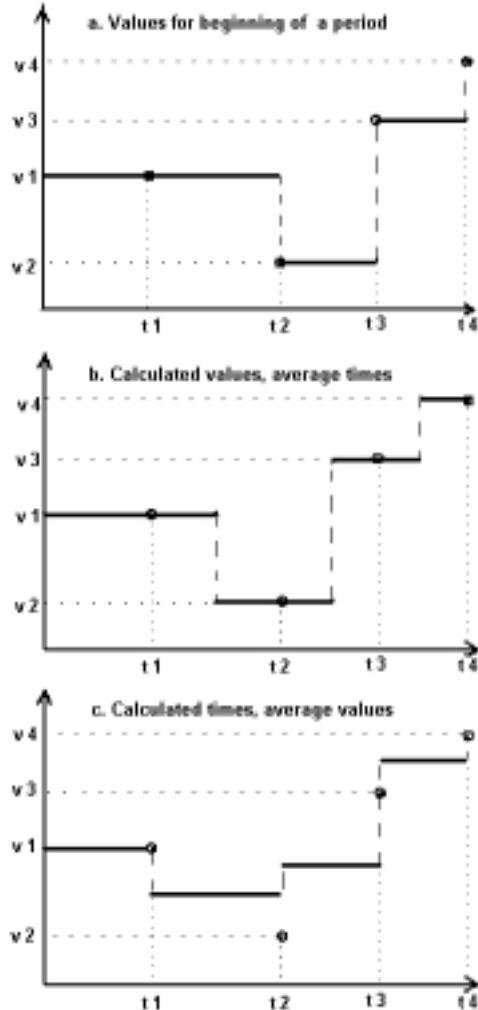


Information about the file including the number of items, minimum and maximum values, and units used will appear in the **Information** area.

Next you have to select the method of step-wise approximation.

Selecting the method of step-wise approximation

The figure on the following page illustrates different ways of step-wise approximation for time-dependent data. In the given example, values **v1**, **v2**, **v3** and **v4** have been simulated by the HELP model to occur at times **t1**, **t2**, **t3** and **t4**. Monthly output at times **t1**, **t2**, **t3** and **t4** represent the middle of months 1, 2, 3 and 4.



On the graph, thick horizontal lines illustrate how the output variable will be represented in the export file. If the options **Values for beginning of a period**, **Calculated values, average times** and **Calculated times, average values** are selected, the line format of the output file is as follows:

time started - delimiter - time ended - delimiter - value.

If **No approximation** is selected, the line format of the export file is:

time simulated - delimiter - value.

After examining the previous explanations, you may finish importing data from the HELP model:

- 1) the drop-down arrow beside the **Stepwise approximation** drop-down listbox and select the appropriate approximation method (e.g. **Values for beginning of a period**),
- 2) **Calculate intensity** check-box. This will divide the monthly totals for percolation through the ash layer by 30.4, which is the daily average precipitation.

Note: this is requested because the HELP model produces recharge values accumulated over a selected time interval (day, month or year) in units of length ('m' in Metric unit template) while VS2DT requires input in units of flux (m/day). This functions should not be activated if the HELP output was produced with daily intervals.

Note: If the percolation data were accumulated over a year (Annual Output in Visual HELP) the results will be divided by 365.25 when this function is applied.

3) Import

The imported data will appear as the **Flow Upper Boundary** parameter group:

The screenshot shows a Windows dialog box titled "Edit Parameters" with the sub-section "VS2DT|Flow Upper Boundary". The table lists 20 rows of data, each representing a boundary condition. The columns are: Row#, Start Time (day), End Time (day), Boundary Type, Units, Value, and Allowed Pending (checkbox). The data shows a series of flux values starting at 0.000E+00 and increasing to 3.1434E-04. The "Allowed Pending" column contains several checked checkboxes.

Row#	Start Time day	End Time day	Boundary Type	Units	Value	Allowed Pending
1 (30.416)	0.000	60.033	Flux	m/day	0.0000E+00	<input checked="" type="checkbox"/>
2 (30.417)	60.033	91.250	Flux	m/day	0.0000E+00	<input checked="" type="checkbox"/>
3 (30.417)	91.250	121.667	Flux	m/day	0.0000E+00	<input checked="" type="checkbox"/>
4 (30.417)	121.667	152.083	Flux	m/day	2.2995E-03	<input checked="" type="checkbox"/>
5 (30.417)	152.083	182.500	Flux	m/day	1.2798E-04	<input checked="" type="checkbox"/>
6 (30.417)	182.500	212.917	Flux	m/day	1.0673E-04	<input checked="" type="checkbox"/>
7 (30.416)	212.917	243.333	Flux	m/day	1.2477E-05	<input checked="" type="checkbox"/>
8 (30.417)	243.333	273.750	Flux	m/day	7.8883E-05	<input checked="" type="checkbox"/>
9 (30.417)	273.750	304.167	Flux	m/day	1.5889E-05	<input checked="" type="checkbox"/>
10 (30.416)	304.167	334.583	Flux	m/day	2.6977E-03	<input checked="" type="checkbox"/>
11 (30.417)	334.583	365.000	Flux	m/day	3.4539E-03	<input checked="" type="checkbox"/>
12 (30.417)	365.000	395.417	Flux	m/day	1.5790E-03	<input checked="" type="checkbox"/>
13 (30.416)	395.417	425.833	Flux	m/day	0.0000E+00	<input checked="" type="checkbox"/>
14 (30.417)	425.833	456.250	Flux	m/day	0.0000E+00	<input checked="" type="checkbox"/>
15 (30.417)	456.250	486.667	Flux	m/day	0.0000E+00	<input checked="" type="checkbox"/>
16 (30.416)	486.667	517.083	Flux	m/day	2.1137E-03	<input checked="" type="checkbox"/>
17 (30.417)	517.083	547.500	Flux	m/day	5.4314E-05	<input checked="" type="checkbox"/>
18 (30.417)	547.500	577.917	Flux	m/day	4.3462E-05	<input checked="" type="checkbox"/>
19 (30.416)	577.917	608.333	Flux	m/day	3.1434E-04	<input checked="" type="checkbox"/>
20 (30.417)	608.333	638.750	Flux	m/day	1.7400E-01	<input checked="" type="checkbox"/>

Now, you may specify the rest of the project settings and run the VS2DT model with accurate data for the flow upper boundary which was created with the HELP simulation.

Export from WHI UnSat Suite

Prior to exporting data from an WHI UnSat Suite model, ensure the appropriate input data is used with the model. In this section, exporting HELP and VS2DT results for use as input into Visual MODFLOW is described. These two cases covers almost all available variants.

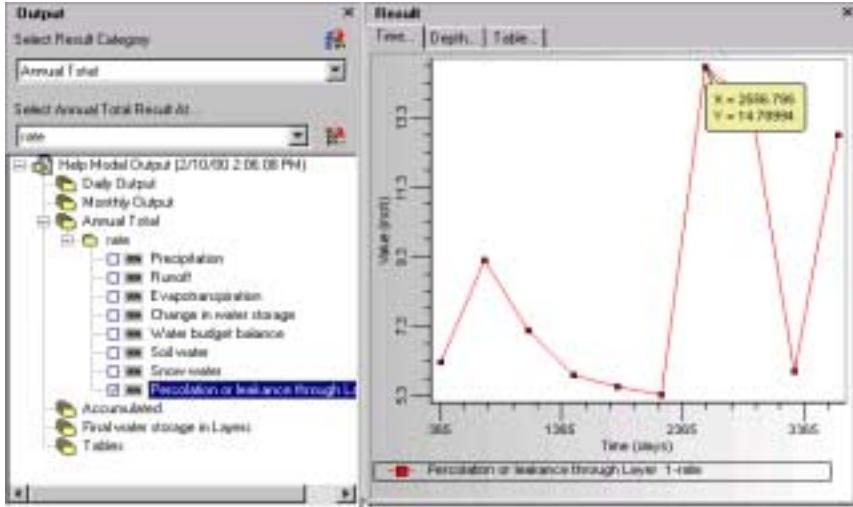
Export of Visual HELP data to Visual MODFLOW

An essential input variable that can be effectively assessed by Visual HELP is groundwater recharge. Visual HELP can produce this variable for daily, monthly, or annual intervals. To help illustrate this feature, a sample project using the settings from the Visual HELP Infiltration Lab can be used (refer to the included lab). To simplify matters, only 10 years of simulated data is used, as opposed to 100 years in the lab. For demonstrating of the data export feature, let us use monthly time intervals to approximate recharge with recharge rate units required by Visual MODFLOW be ‘m/day’.

Both WHI UnSat Suite and Visual MODFLOW allow the user to select from a number of different time scales. However, the Visual MODFLOW allows you to select the units for recharge rate separately, while the Visual HELP uses consistent units for time and length for all output results. In addition, the HELP model produces recharge values (percolation through the unsaturated zone) accumulated over a selected time interval (day, month or year) in units of length while Visual MODFLOW requires input in units of flux (m/day). As such, you must be careful to choose your recharge rate in Visual MODFLOW such that the units of length in the numerator and the units of time in the denominator (e.g. m/day) are the same as the units of time in the Visual HELP model. For example, if the Visual HELP model length units are ‘m’ and time units are ‘days’, you must select units of recharge in ‘m/day’.

However, if you wish to have your recharge to be imported into the Visual MODFLOW in specific units or the Visual HELP units does not match with allowed Visual MODFLOW recharge units, you need to change length and time units in the Visual HELP model.

As a result of running the Visual HELP Infiltration Lab, and subsequently selecting the **Annual Total/rate/ Percolation or leakance through Layer 1** option in the Output tree, the following figure appears:



The graph in the right window shows that peak recharge occurs in year 7 of the simulation with an annual recharge rate of approximately 14.8 inches.

Next, you will learn how to convert this data into the required format and export to Visual MODFLOW.

To export data from UnSat Suite, the following steps need to be executed:

- i) Set output units in Visual HELP equal to the desired Visual MODFLOW units.
- ii) Open the Visual HELP output with the desired time intervals
- iii) Export the data.

Setting output units in Visual HELP equal to the desired Visual MODFLOW units

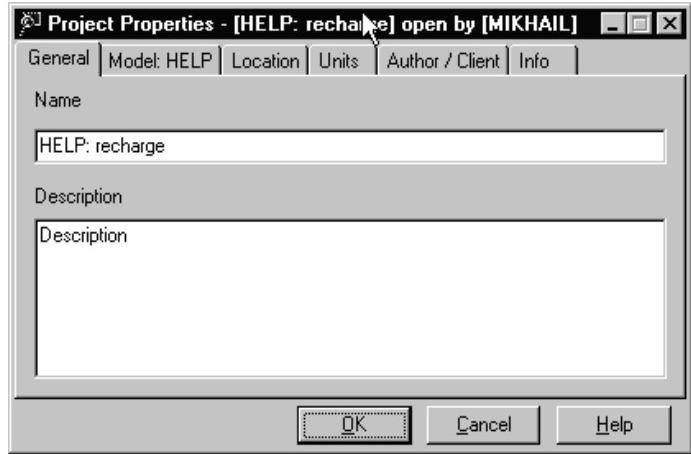
Visual MODFLOW requires input in units of recharge rate (m/day) averaged over monthly time intervals. Because the units for recharge in the quoted Visual HELP Lab exercise are ‘in’ totalled over a year intervals, units in HELP output have to be corrected.

To change units, in the Main menu:

☞ **Project**

☞ **Properties**

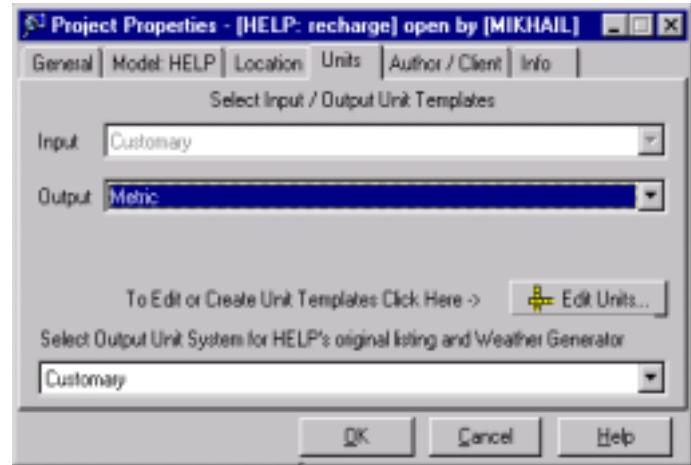
The following dialog box will appear:



☞ **Units** tab

The **Units** selection dialog box will appear.

- ☞ the drop-down arrow beside **Output** and select **Metric** from the drop-down list:



The metric Unit template has unit 'm' for length and 'days' for time.

- ☞ **OK** to close the dialog box.

Note: the example above shows how to change units in a previously created project when units differ from the desired units. If you are creating a new project for HELP simulation, set appropriate units right at the project set-up (see Chapter 2).

Opening the Visual HELP output with the desired time intervals

To open the recharge data with monthly time intervals,

- ☞ the drop-down arrow in the **Select Result category** drop-down list box and then

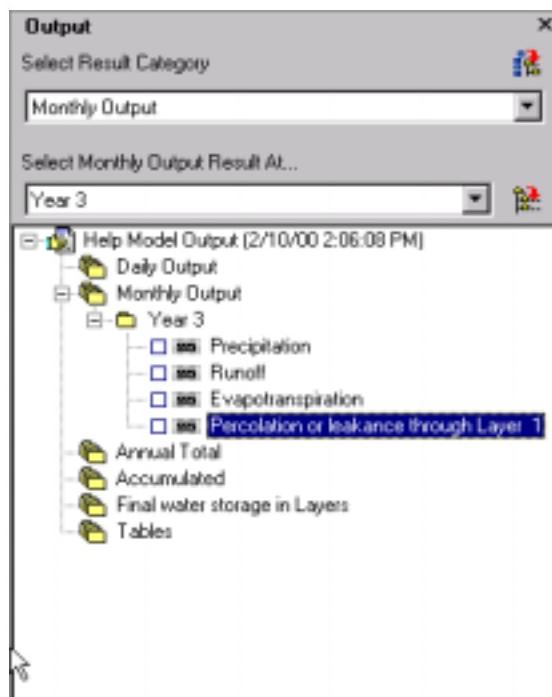
☞ Monthly Output

- ☞ **Year 3** in the **Select Monthly Output Result at...**drop-down list box.



- ☞ the icon to the right of the **Select Monthly Output Result at...** box.

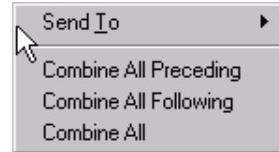
The list of available monthly variables for year 3 will open in the Result Tree:



Now select the check-box beside the **Percolation or leakage through Layer 1** in the Output Tree, as it is used in the WHI UnSat Suite to select variables. You will produce a graph of percolation through the bottom of the unsaturated profile (the recharge rate) for year 3 in the **Result** window.

However, the task is to export recharge data for all 10 years of simulation. To do this,

click the right mouse button at the name of the variable **Percolation or leakage through Layer 1**. The following dialog box will appear:



This dialog box allows you to select the data aggregation method.

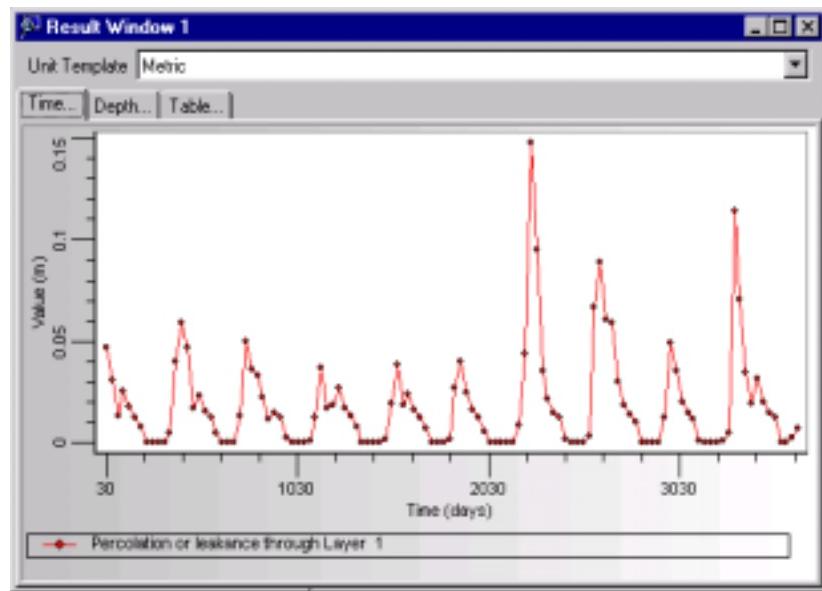
If you select **Combine All Preceding**, the data for all preceding years and the current year will be presented.

If you select **Combine All Following**, the data for the current year and all following years will be presented.

If you select **Combine All**, the data for the whole simulation period will be presented.

Combine All

The following graph will appear in the **Result Window 1**:



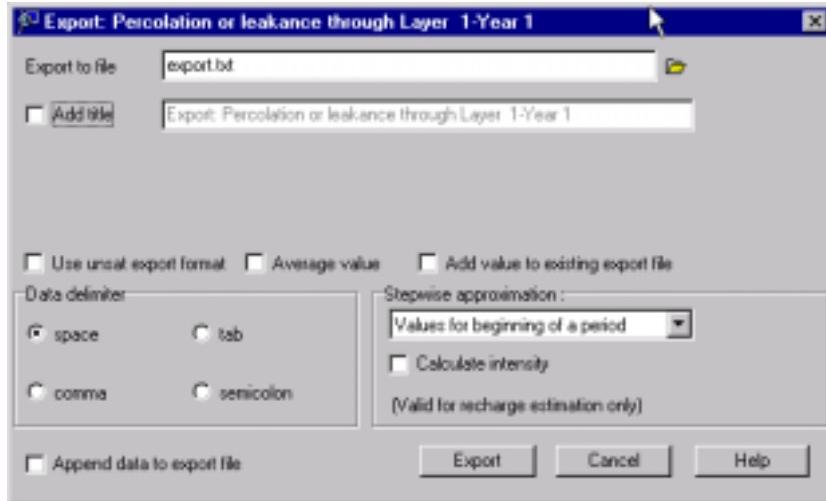
This graph shows monthly variation of recharge over the 10-year period.

Exporting data

To export data presented on the graph, place the mouse pointer at the graph line (the bubble sign will appear) and click right mouse button. A button named **Export..** will appear.

Export..

The **Export:Percolation or leakance through Layer 1** dialog box will appear:



This dialog box allows you to specify the following settings for data export:

- the step-wise approximation method
- the data delimiter type
- the export file destination

To export recharge rate from the Visual HELP model:

1) Select the step-wise approximation method. To do this,

☞ the drop-down arrow beside the **Stepwise approximation** drop-down listbox and select the appropriate method of approximation (e.g. **Values for beginning of a period**).

Note: To learn about step-wise approximation methods, see the explanation in the previous section (p. 296).

2) ☞ **Calculate intensity** check-box. This will divide the monthly totals for precipitation by 30.4, to calculate the average daily recharge (recharge rate).

Note: If the percolation data were accumulated over a year (Annual Output in Visual HELP) the results will be divided by 365.25 when this function is applied.

Note: this function should not be activated if the HELP output was produced with daily intervals and you use units 'ft/day' or 'm/

day' for recharge rate in Visual MODFLOW. The same applies to the case when Annual Total values are used for Recharge in HELP and 'mm/year' or 'in/year' - for recharge rate in Visual MODFLOW.

3) Edit the file name in the **Export to file** text box, if necessary.

4)  the icon to the right of the **Export to file** text box. This will activate the browser. Navigate to the directory where you would like to store your output file.

5)  the **Export** button.

The export data file will appear in the designated directory. A sample fragment of the recharge file is shown below.



```
0 60 0.333 0.00153943
60.333 91.25 0.00101395
91.25 121.667 0.000441983
121.667 152.083 0.000850885
152.083 182.5 0.000587345
182.5 212.917 0.000418009
212.917 243.333 0.000271938
243.333 273.75 0
273.75 304.167 0
304.167 334.583 0
334.583 365 2.46225E-06
365 395.417 0.000169818
395.417 425.833 0.00132375
425.833 456.25 0.00195165
456.25 486.667 0.00152974
486.667 517.083 0.000551313
517.083 547.5 0.000759203
547.5 577.917 0.000519504
577.917 608.333 0.000403672
```

Note: In Visual MODFLOW, the recharge data saved in this file can be imported at the following part of the interface: Input/Boundaries/Recharge/Edit/Property.

Export of SESOIL data to Visual MODFLOW

The SESOIL model can be effectively used for assessment of both groundwater recharge and contaminant inflow concentration at the groundwater surface. As it is known, SESOIL is not the only model within the WHI UnSat Suite which may be used for this purpose.

Particularly, Visual HELP with its well developed routine for simulation of soil-atmospheric boundary processes may produce the most reliable

assessments of groundwater recharge. In its turn, VS2DT which utilizes the Richard's equation for moisture transport and enables simulating variety of chemical processes, may produce a very reliable estimate of contaminant inflow at the groundwater surface. However, SESOIL allows assessment of both variables within the same model in a most efficient and well information-balanced way. Taking into account the quality of the input data commonly available, we would recommend using SESOIL (in conjunction with the Weather Generator) for assessment of the recharge and contaminant inflow at the groundwater surface in majority of cases. In addition, the capability of SESOIL to simulate hydrolysis and metal complexation makes it the only tool in some specific cases.

This section explains how to convert SESOIL results for groundwater recharge and contaminant inflow concentration into the required format and export to Visual MODFLOW or to other program. Export of other variables, if required, may be executed in the same manner. To explain the technology, the project case from the SESOIL Demo Lab is used.

To export data from SESOIL, the following steps need to be executed:

- i) Open the SESOIL output with the desired Visual MODFLOW time units,
- ii) Select variables to export and combine data if monthly data is desired;
- iii) Export the data.

Open the SESOIL output with the desired Visual MODFLOW time units

Both WHI UnSat Suite and Visual MODFLOW allow the user to select from a number of different time scales. However, the Visual MODFLOW allows you to select the units for recharge rate separately, while the WHI Suite uses consistent units for time and length for all output results. As such, you must be careful to choose your recharge rate in Visual MODFLOW such that the units of length in the numerator and the units of time in the denominator (e.g. m/day) are the same as the units of length and time in the SESOIL model. For example, if the SESOIL model length units are 'm' and time units are 'days', you must select in Visual MODFLOW 'm/day' units for recharge.

However, if existing units of SESOIL model do not allow you to select appropriate Visual MODFLOW recharge units, you have to change units in the SESOIL model.

Note: There is no need to change units in SESOIL if your project utilizes the default Customary or Metric unit templates for the output. The Customary template uses 'feet' units for length and Metric template uses 'm' for

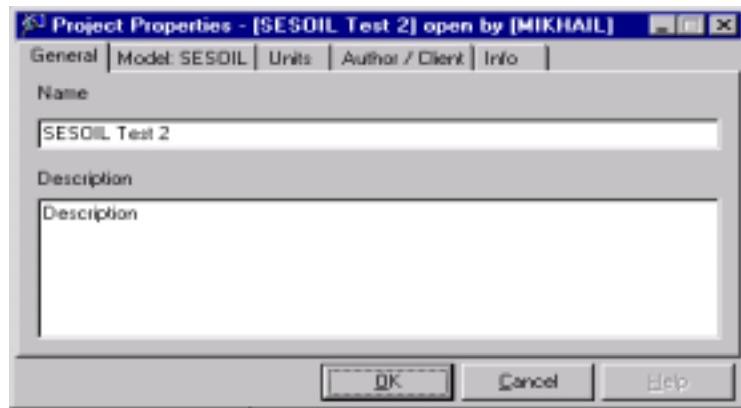
length. Both templates use ‘days’ units for time. In these cases you may use units ‘ft/day’ or ‘m/day’ for recharge rate in Visual MODFLOW.

To change units in the SESOIL model, in the Main menu:

☞ **Project**

☞ **Properties**

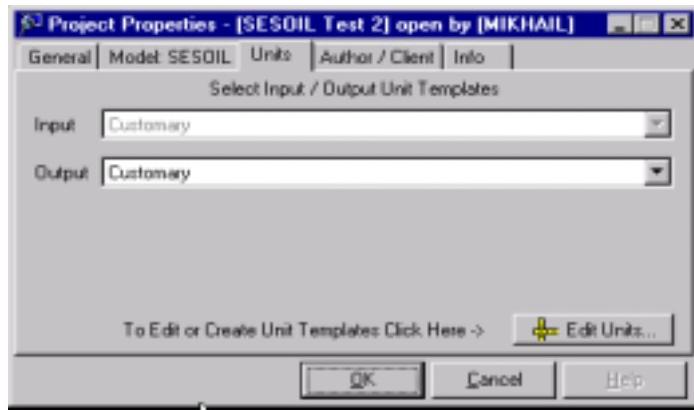
The following dialog box will appear:



☞ **Units tab**

The **Units** selection dialog box will appear.

☞ the drop-down arrow beside **Output** and select **Customary** or **Metric** from the drop-down list:



☞ the **Edit Units** button to view/edit the model units. Please refer to section 2 for a detail description of unit templates.

☞ **OK** to close the dialog box.

Select variables to export, and combine data (monthly output)

As it was explained in Chapter 15, the SESOIL model calculates unsaturated flow and contaminant transport using monthly intervals. In the output the variation of groundwater recharge and contaminant concentration in recharge are presented in two time scales:

- annually in result category **Specified Depth (Annual Summary)**,
- monthly in result category **Specified Depth (Monthly Output)**,

Monthly output data are presented in 12-number sets (one number per month) and the output contains **N** such sets where **N** is the number of years simulated. To export the monthly data for each year of simulation, you must combine all annual data sets for selected variable.

Next you will learn how to select annual data for groundwater recharge and contaminant concentration.

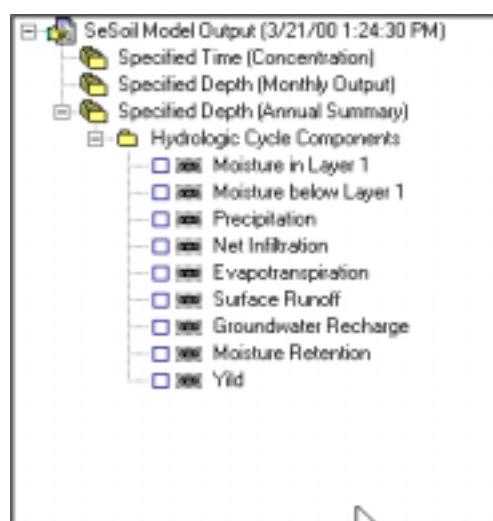
Selecting annual groundwater recharge

☞ **Specified Depth (Annual Summary)** in the **Select Result Category** drop-down list box.

The lower drop-down list box **Select Specified Depth (Annual Summary) Result at...** will show the first available sub-category **Hydrologic Cycle Components**.

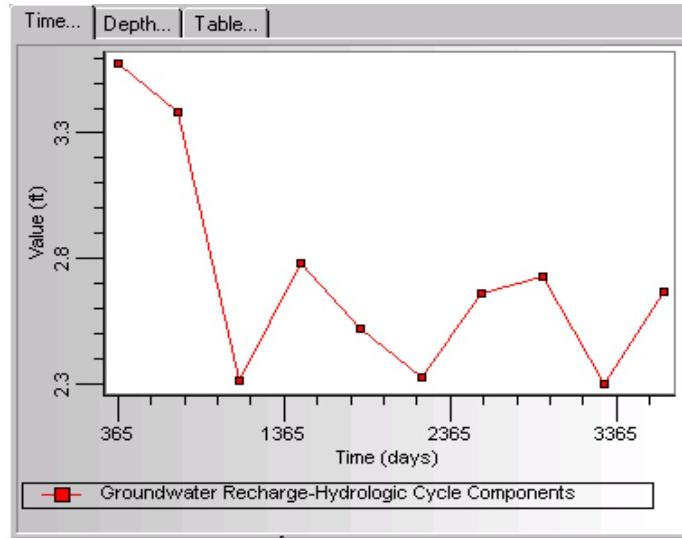
☞ the icon  to the right of the **Select Depth (Annual Summary) Result at...** box.

The list of available hydrologic variables will appear:



☞ the check-boxes beside **Groundwater Recharge**.

A graph for Groundwater Recharge will appear in the Result View window:



Now you may examine simulated Recharge. The steps to export the Groundwater Recharge rates are described later in the section.

Next you will learn how to create a graph of the annual contaminant concentration in recharge.

Selecting annual contaminant concentration in recharge

☞ the arrow in the **Select Specified Depth (Annual Summary) Result at...** window

The list of available sub-categories will appear:



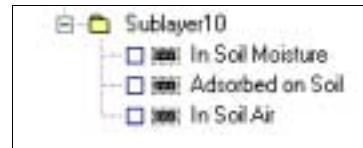
Pollutant Concentration.

☞ the icon  to the right of the **Select Depth (Annual Summary) Result at...** box.

A list of four profile layers will appear in the Output Tree. You may need to use the side slider in the Output Tree to display this sub-category.:

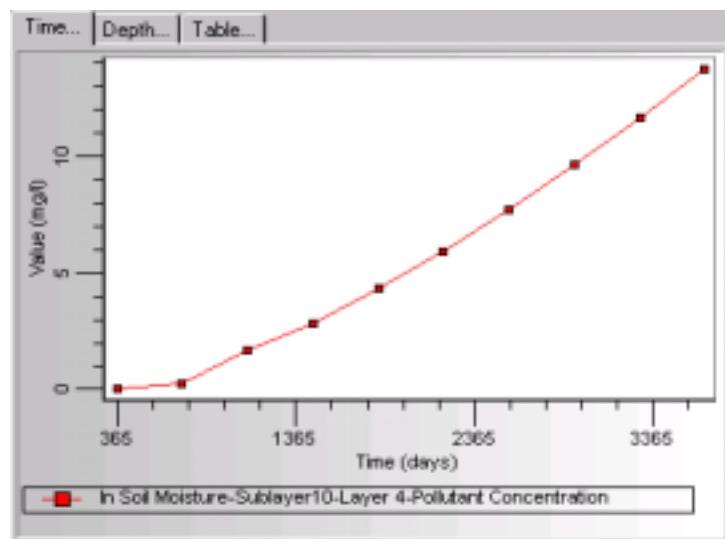


- ☞ '+' sign to the left of **Layer 4** (10 sub-layers will appear in the tree)
- ☞ '+' sign to the left of **Sub-layer 10** (available variables will appear in the tree)



- ☞ the check box beside **In Soil Moisture**

The graph for concentration at Sub-layer 10, Layer 4, will appear in the **Output** View window. This is the point closest to the groundwater surface:



Note: If you have specified less than 4 layers within the unsaturated profile, open the bottom sublayer of the lowest layer in the profile to reach contaminant concentration in recharge.

Next you will learn how to prepare monthly groundwater recharge summaries.

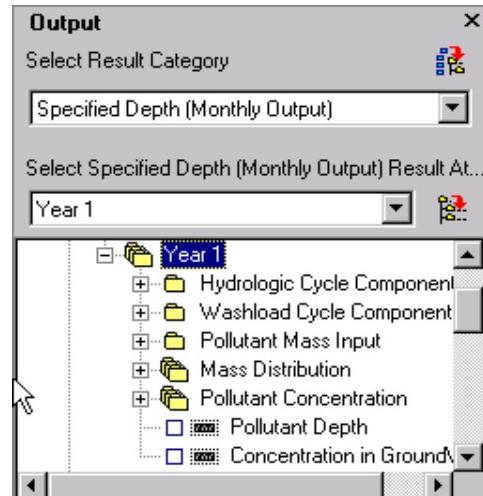
Preparing graph of the monthly groundwater recharge

☞ **Specified Depth (Monthly Output)** (in the **Select Result Category** drop-down list)

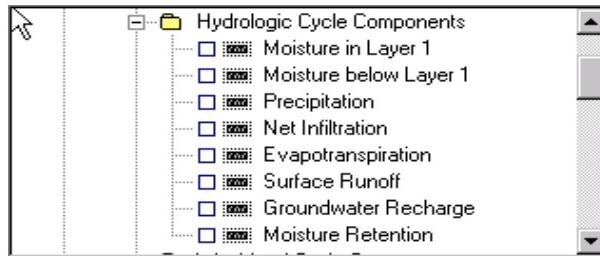
The dialogue window will display the first available annual set of results.

☞ the icon  to the right of the **Select Specified Depth (Monthly Output) Result at...** box.

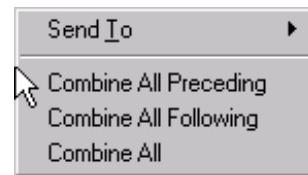
The list of available monthly variables for year 1 will open in the Result Tree:



☞ the ‘+’ sign to the right of the **Hydrologic Cycle Components**:



Right click **Groundwater Recharge**. The following menu box will appear:



This dialog box allows you to select the data aggregation method.

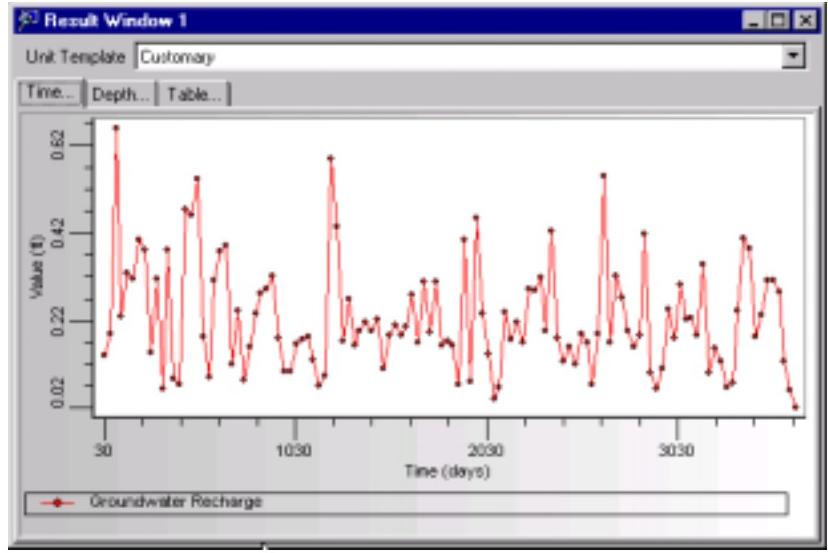
If you select **Combine All Preceding**, the data for all preceding years and the current year will be presented.

If you select **Combine All Following**, the data for the current year and all following years will be presented.

If you select **Combine All**, the data for the whole simulation period will be presented.

☞ **Combine All**

After a short computation reflected with a progress bar, the following graph will appear in the separate window:



This graph presents all monthly sets of Groundwater Recharge combined in one set.

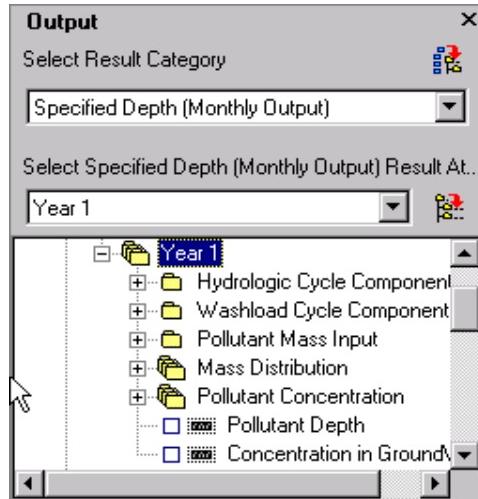
Preparing graph of the monthly concentration of contaminant in groundwater recharge

☞ **Specified Depth (Monthly Output)** (in the **Select Result Category** drop-down list)

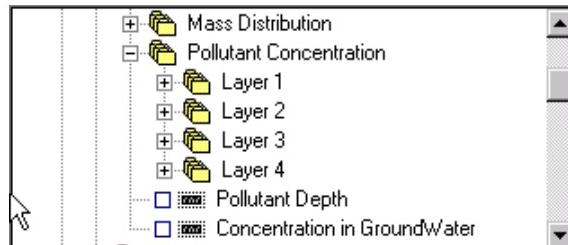
The dialogue window will display the first available annual set of results.

☞ the icon  to the right of the **Select Specified Depth (Monthly Output) Result at...** box.

The list of available monthly variables for year 1 will open in the Result Tree:



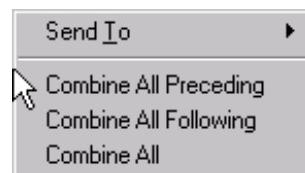
☞ the '+' sign to the right of the **Pollutant Concentration**. The list of four profile layers will appear in the Output Tree.



☞ the '+' sign to the left of **Layer 4**. Ten sublayers will appear in the tree.

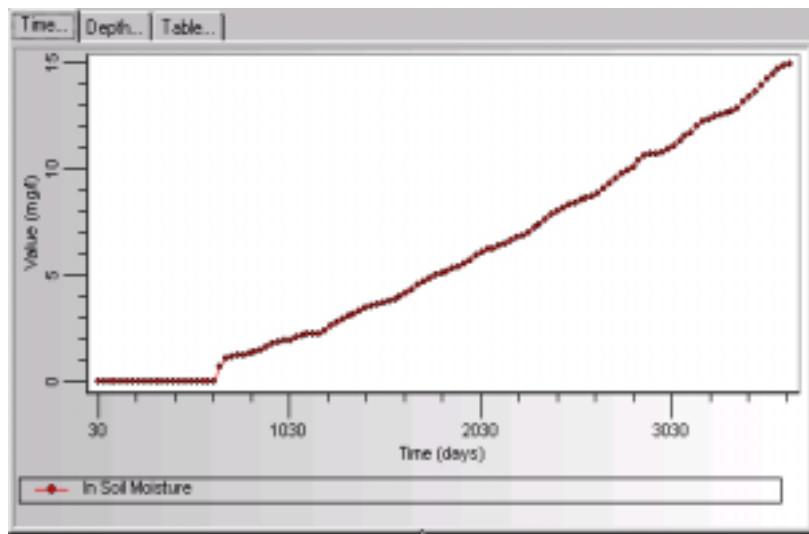
☞ the '+' sign to the left of **Sublayer 10**. Available variables for this sublayer will appear in the tree.

Right click **In Soil Moisture**. The menu box will appear:



☞ **Combine All**

After a short computation reflected with a progress bar, the following graph will appear in the separate window:



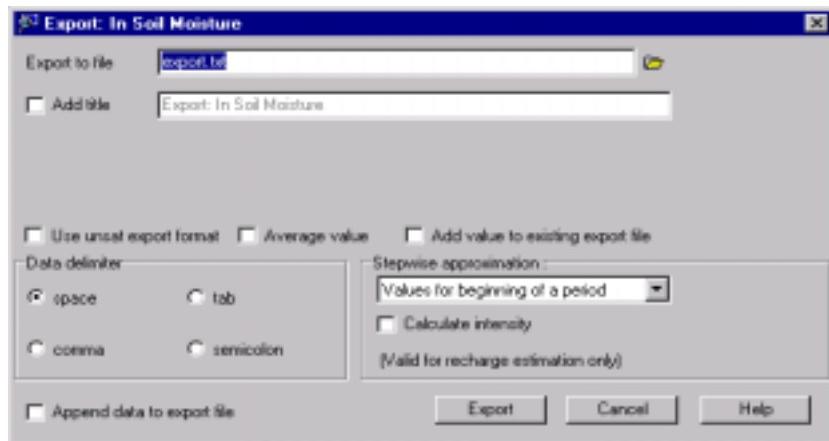
The graph for concentration at Sublayer 10, Layer 4, the point closest to the groundwater surface, will appear in the **Output** View window combined for the whole simulated period.

Exporting data

To export data presented on the graph, place the mouse pointer at the graph line (the bubble sign will appear) and click right mouse button. A button named **Export..** will appear.

☞ Export..

The **Export: [Variable Name]** dialog box will appear:



This dialog box allows you to specify the following settings for data export:

- the step-wise approximation method
- the data delimiter type
- the export file destination

To export recharge rate from the SESOIL model:

1) Select the step-wise approximation method. To do this,

☞ the drop-down arrow beside the **Stepwise approximation** drop-down listbox and select the appropriate method of approximation (e.g. **Values for beginning of a period**).

Note: To learn about step-wise approximation methods, see the explanation in the previous section (p. 296).

2) ☞ **Calculate intensity** check-box. This will divide the monthly totals for precipitation by 30.4, to calculate the daily average recharge (the recharge rate).

3) Edit the file name in the **Export to file** text box, if necessary.

4) ☞ the icon  to the right of the **Export to file** text box. This will activate the browser. Navigate to the directory where you would like to store your output file.

5) ☞ the **Export** button.

The export data for recharge file will appear in the designated directory. A sample fragment of the recharge file is shown below.

```
0 60.8333 0.00454354
60.8333 91.25 0.00616902
91.25 121.667 0.0216221
121.667 152.083 0.00753222
152.083 182.5 0.0107337
182.5 212.917 0.0103651
212.917 243.333 0.0132925
243.333 273.75 0.0124725
273.75 304.167 0.00479261
304.167 334.583 0.0102644
334.583 365 0.00204997
```

Note:

Note: In Visual MODFLOW, the recharge data saved in this file can be imported at the

following part of the interface: Input/Boundaries/Recharge/Edit/Property.

The export contaminant concentration in recharge, the sequence of actions should be similar to the previously described for recharge rate, except the check box **Calculate intensity** should not be activated.

A sample fragment of the recharge concentration file is shown below.

```
.....  
365 395.417 0  
395.417 425.833 0  
425.833 456.25 0  
456.25 486.667 0  
486.667 517.083 0  
517.083 547.5 0  
547.5 577.917 0  
577.917 608.333 0  
608.333 638.75 0  
638.75 669.167 0  
669.167 699.583 0.712375  
699.583 730 1.10469  
730 760.417 1.18396  
760.417 790.833 1.24038  
790.833 821.25 1.27719  
821.25 851.667 1.32578  
851.667 882.083 1.41554
```

In this file the first number in a row shows the beginning of the time interval in days, the second number is the end of time interval in days and the third number is concentration of contaminant in recharge in mg/l.

Export of VS2DT data to Visual MODFLOW

To export results from an unsaturated transport simulation obtained with the VS2DT model, including the multi-species results, the same tools that were described above should be used.

The following steps need to be executed:

- i) Create the VS2DT project and set output units equal to the desired units of the model-recipient.
- ii) Run the VS2DT model.
- iii) Export data from VS2DT.

The most promising application of the VS2DT model is the possibility to accurately assess pollutant concentrations reaching the groundwater surface, and subsequently use this data as input for Visual MODFLOW.

This section describes tools used for VS2DT export. To learn how to create a VS2DT project, see Chapters 2 and 10.

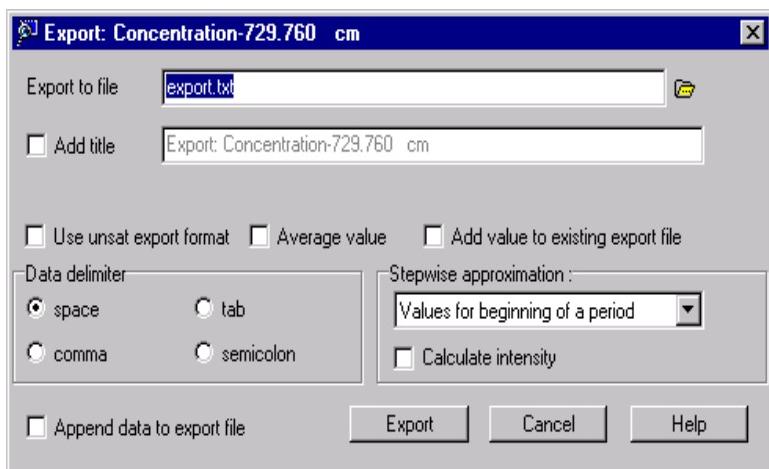
After you create a VS2DT project representing the unsaturated profile and subsequently run the model, select a **Specified Depth** close to the groundwater surface from the list in the **Output** window.

Open all available variables for this depth and select **Concentration**. The breakthrough curve for concentration at the specified depth will appear in the **Result** window.

To export data presented on the graph, place the mouse pointer at the graph line (the bubble sign will appear) and click right mouse button. A button named **Export..** will appear.

Export..

The same Export dialog box that was described above for the Visual HELP export will appear.



This dialog box allows you to specify the following settings for data export:

- the step-wise approximation method
- the data delimiter type
- the export file destination

In addition, you may:

- add a title to the file by selecting **Add title**,
- calculate the average value by selecting **Average value**

Note: do not activate the Calculate intensity check-box while exporting Concentration data.

For more details about export, see previous sections of this chapter. When you are finished setting parameters for the export,

☞ **Export** button.

Export of VS2DT data to Visual MODFLOW for multiple species

As stated in the VS2DT description (Part 4, Introduction), this model is capable of simulating transport for only one chemical at a time. However, you can produce an export file that will contain more than one species.

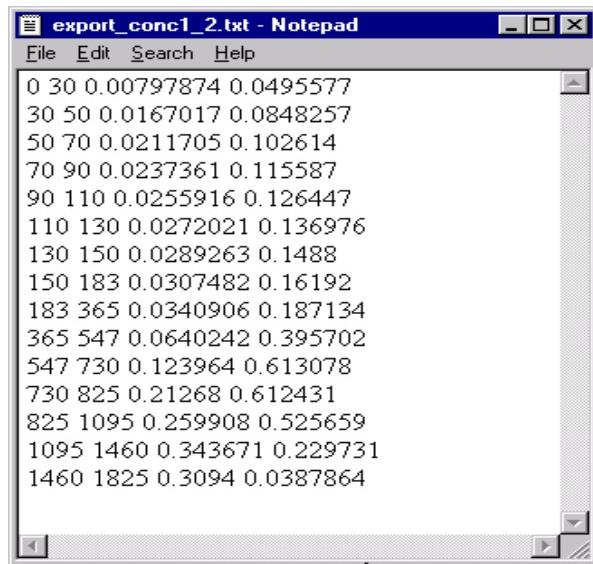
The following algorithm allows you to create such a file:

- determine the sequence of the species in the export file,
- run the VS2DT model for the first species,
- create the export file for this species as described in the previous section,
- set new transport boundaries and initial chemical conditions for the second species while retaining the flow settings,
- run the VS2DT model for the second species,
- open the **Export** dialog box.

In the **Export** dialog box:

- set export settings for the second species similar to the settings for the first species,
- in the **Export to file** text box select the file to which the first species was saved,
- select **Add value to existing export file** check-box.

Concentration values for the second species will appear in the column next to the first species.



The screenshot shows a Windows Notepad window with the title bar 'export_conc1_2.txt - Notepad'. The menu bar includes 'File', 'Edit', 'Search', and 'Help'. The main window displays a list of numerical values separated by spaces, representing concentration data. The data consists of 15 rows, each containing four values: time, concentration of the first species, concentration of the second species, and possibly a third or fourth value. The values range from approximately 0 to 1460.

	Time	Conc1	Conc2	Conc3
1	0	0.00797874	0.0495577	
2	30	0.0167017	0.0848257	
3	50	0.0211705	0.102614	
4	70	0.0237361	0.115587	
5	90	0.0255916	0.126447	
6	110	0.0272021	0.136976	
7	130	0.0289263	0.1488	
8	150	0.0307482	0.16192	
9	183	0.0340906	0.187134	
10	365	0.0640242	0.395702	
11	547	0.123964	0.613078	
12	730	0.21268	0.612431	
13	825	0.259908	0.525659	
14	1095	0.343671	0.229731	
15	1460	0.3094	0.0387864	

After adding the second species, you may add as many as required by your simulation scenario using the same algorithm.

Index

A

Accumulated Volumes 136
acknowledgements 8
add new contaminant 35
add observation point 159
Adding new contaminant to the database 35
Africa 104
air diffusion 235
AIRFLOW/SVE 8
anisotropy 211
ANNUAL INPUT table 261
Annual Total 136
AquiferTest 8
archive project set 63
area 155, 204, 236, 274
Asia 104
atmospheric pressure 186
Australia 104
authors 26

B

balance result 167, 245, 294
balance summary 194
barrier soil liners 77, 83
bottom elevation 29
boundary conditions 186
bulk density 158, 239, 275

C

Canada 104
Canadian Climate Centre 101
Canadian Climate Centre format 123
case settings 79, 149, 175, 231, 255
Case Settings group 41
case specifications 175
cell number 231, 257, 261
characteristic curve coefficient 158
Chemical 38
clients 26
CN 80
company 26, 27
concentration distribution
 initial condition 201
concentration in recharge water 232
consulting 8
contact person 27
contaminant
 properties 235, 268
 specify 232, 258
 substitute 234, 273
Contaminant Load Schedule 271
Contouring Factor 266
conventions
 VS2DT 188
co-ordinates 20
Copy Holder icon 16

copy project 67
Create New Profile icon 15
Create New Project icon 15
create report 169, 224
Creating a New Profile Using a Profile Template 30
curve number 80
customary units 23

D

database
 weather generator 120
defaults
 project set 64
 stress period 192
Delete Layers icon 16
delete observation point 161
delete project set 63
dependent soil parameters 212
dependent transport parameters 215
depth 164, 242, 291, 294
description 18
Design Layer Permissions 34
Designing profile 29
dispersion coefficient 158
DOS HELP Output 134
drainage layers 77, 83
drained 90

E

edit soil properties 157, 238, 276
effective porosity 239
elevation 29, 155, 204, 236, 274
Europe 104
evaporation 99
evapotranspiration
 case setting 180
 parameters 183
 tab 103
Exit icon 16
Export from WHI UnSat Suite 299
Export of SESOIL data 315
Export of Visual HELP data to Visual
 MODFLOW 309
Export of VS2DT data to Visual MODFLOW 327

F

field capacity 83
File menu item 14
finite difference grid 195
FLONET/TRANS 7
flow boundaries 188
flow equation closure 181
FLOWPATH 7
former USSR 104
fraction organic content 158
free air diffusion 235

G

general features 13

generate
 weather 99, 114
generator
 weather 144
geomembrane liners 77, 83
geonets 77, 83
geotextiles 77, 83
GIS map 20, 108
GIS searcher 22, 108
graph
 edit 56
 print 58
 view 51, 163, 218, 242, 281
Graphs 142
grid
 finite difference 195
groundwater recharge 309, 315, 316, 318
group 26

H

hardware requirements 11
head change 192
head criterion 192
Help menu item 15
HELP model 71
 run 134
HELP Output 134
Help Topics icon 16
Henry's law 235
historical data 103
history 71
hydraulic conductivity 211
hydrologic parameters 210

I

icons
 operational 15
 profile 16
import weather data 123, 124
indicator of status 39
infiltration 99
initial conditions
 chemical distribution 179
 set 200
 VLEACH 232, 265
 water 179
initial moisture settings 81
initial surface water 82
initial time step 192
insert layer 89
installation 11
installation defects 84
interface 13
INTERPRETING OUTPUT 133
Intrinsic Permeability 277
introduction 3
iterations
 maximum 181
 minimum 181

K

key contact person 27

L

landfill profile 75
lateral drainage layers 77, 83
latitude 20, 22
layer
 delete 91
 group 96
 initial condition 201
 insert 89
 merge 45, 205
 properties 83, 209
 resize 49, 87, 155, 208, 237, 276
 restore 47, 93, 206
 rules 78
 split 47, 94, 207
 structure 204
 substitute 156, 237, 275
Layer Material 38
leaves 75
length
 simulation 133, 149, 231, 257
license agreement iii
liners 77, 83
load project 28
location 19, 22
longitude 20, 22

M

Mail To icon 16
main menu 14
Manning's Coefficient 267
map icon 102
map network project set 65
Markov Chain model 99
material
 category 90
 substitute 210
material category 31
Material Designer 32
material designer 32
Materials in Profile 38
mean air temperature 99
menu
 main 14
 merge layers 45, 205
Meteo station 110
meteorological station 99
metric units 23
model 18
model stress periods 225
moisture content 84
moisture settings
 initial 81
multiplier 192, 195

N

name 18
Natural Profile 29
new features 4, 72
new profile 30
new profile template 32
new project 17
NOAA format 124
North America 104
notation 13

O

observation points 38, 159, 167, 197
observation times 152, 193
Open Project icon 15
operational icons 15
Operations with the Objects of Profile Structure 39
organic carbon 235, 269
organic content 239
organization 27
Original DOS HELP Output 134
original DOS PESTAN output 163
original DOS VLEACH output 241, 279
Original Listing 135, 137
original listing 163, 241, 279
output
 graphs 163, 218, 242, 281
Output menu item 15
output times 193
output view 17

P

parameters
 evapotranspiration 183
 hydrologic 210
 soil 157
 transport 212
percolation 75, 82
period 185
permanent soil parameters 211
permanent transport parameters 215
pinhole density 84
placement quality 84
plants 75
porosity 83, 211
potential evaporation rate 186
potential evapotranspiration rate 186
precipitation 75, 99, 114
Prepare a Report icon 16
PRINCE 8
print graph 58
Print Preview icon 17
profile
 edit 203
 icons 16
 landfill 75
 modify 155
 multiple 49
 new 30
 parameter groups 38

properties 78, 155, 203, 235, 273
template, new 32
view 17, 44
Profile menu item 15
Profile Properties icon 17
profile template 30
project
 archive set 63
 copy 67
 delete set 63
 information 18
 load 28
 new 17
 set 61
 tree 17, 28, 37
Project menu item 14
properties
 contaminant 235, 268
 layer 83, 209
 observation point 160
 profile 78, 155, 203, 235, 273
 soil 157, 238, 276

R

rain 99
rate
 waste application 151
recharge rate 149, 231
reduction factor 192
relaxation 181
Remote Data Access icon 16
Repairing 68
Repairing the Project Set 68
report
 create 169, 224
 prepare 59, 248, 294
requirements
 system 11
residual moisture content 211
resize layer 49, 87, 155, 208, 237, 276
Restore icon 16
restore layer 47, 93, 206
restore observation point 161
Result Category 52
Results View 17
results view 17
root activity 186
root depth 186
root pressure 186
run current model for all profiles 15
Run menu item 15
Run Model For Profile icon 17
run VS2DT model 217
Run Weather Generator icon 15
Running the Model 50
runoff 75, 99
runoff area 82
runoff method 80

S

saturated hydraulic conductivity 83, 158
saturated water content 158
Save Project icon 15
schedule
 waste application 151
search 104
select location 20
Selecting the method of step-wise approximation 306
settings
 case 79, 149, 175
 solver 181
Settings menu item 15
simulation
 length 133, 149, 231, 257
 maximum time 180
 set time 144
 start 32
site 20
slope 90
snow 75, 99
snowmelt 99
software requirements 11
Soil Erodibility Factor 266
soil hydraulics 178
soil hydrologic parameters 210
soil liners 77, 83
Soil Loss Ratio 266
soil parameters 32
soil properties 157, 238, 276
solar radiation 20, 99, 114
solver settings 181
sorption constant 149, 158
South America 104
space differencing 181
specific storage 211
specified time 164, 242, 291, 294
split layer 47, 94, 207
start at bottom 195
start new project 17
start simulation 32
start UnSat Suite 12
start weather generator 101
status indicator 39
step length 195
stress period defaults 192
stress periods 225
sublayers 255
substitute contaminant 234, 273
substitute layer 156, 237, 275
substitute material 210
subsurface inflow 84
surface resistance 186
surface runoff 99
surface storage 99
surface water
 initial 82
 settings 82
symmetric 195
synthetic weather data 103
system requirements 11

T

table
 view 55, 168, 223, 246
Tables 139
temperature 114
template
 units 23
terms 13
text search 104
time
 output 193
 simulation 144
time dependent groups 150
Time Dependent Parameter Group 42
time differencing 181
time steps
 maximum 181
 multiplier 192
 output 232
 simulation 232
top elevation 29
training 8
transmissivity 85
transpiration 99
transport boundaries 190
transport equation closure 181
transport parameters 212
transport simulation 176
tree view 37
two-parameter Gamma distribution model 99

U

unit templates 23
units 23
UnSat Suite profiles 28
US states 104
USSR (former) 104

V

vapor boundaries 232
vegetation class 82
vegetation growth 99
vertical percolation layers 77, 82
View icon 16
View menu item 14
view original files 217
View Original Listing 50
view output graphs 51, 163, 218, 242, 281
View Profile icon 15
view tables 55, 168, 223, 246
view weather data 115
Viewing Visual HELP Tables 139
Visual Groundwater 8
visual HELP model
 run 134
Visual HELP Tables 139
Visual MODFLOW 7
VS2DT conventions 188

W

 Washload Settings 265
 waste application schedule 151
 water
 initial condition 200
 solubility 235, 269
 water content
 saturated 158
 VLEACH 239
 Waterloo Hydrogeologic Inc.
 How to Contact WHI 6
 weather generator 99, 144
 weighted hydr. cond. 181
 wilting point 83
 Window menu item 15
 Working over a Local Area Network 65
 world weather generator database 100

Z

 Zoom In icon 16
 Zoom Out icon 16